ACS SASSI-ANSYS* Integration Capability

Version 3.0
"Options A and AA"

An Advanced Computational Software for 3D Dynamic Analysis Including Soil-Structure Interaction

User Manual
Revision 2

March 31, 2015

Ghiocel Predictive Technologies, Inc.
4 South Main St., 3rd Floor, Pittsford, NY 14534, USA
Phone: (585) 641-0379/ Fax: (585) 586-4672
E-mail: acs.sassi@ghiocel-tech.com

DISCLAIMER

GHIOCEL PREDICTIVE TECHNOLOGIES, INC. DOES NOT WARRANT THE OPERATION OF THE ACS SASSI VERSION 3.0 PROGRAM WILL BE UNINTERUPTED OR ERROR-FREE. GHIOCEL PREDICTIVE TECHNOLOGIES, INC. MAKES NO REPRESENTATIONS OR WARRANTIES, EXPRESS OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTIBILITY AND FITNESS FOR A PARTICULAR PURPOSE. Ghiocel Predictive Technologies, Inc., in any case shall not be liable for any costs, damages, fees, or other liability, nor for any direct, indirect, special, incidental, or consequential damages (including loss of profits) with respect to any claim by LICENSEE or any third party on account of or arising from this License Agreement or use the ACS SASSI Version 3.0 program.

The ACS SASSI Version 3.0 *baseline* code using the standard skyline solver has been extensively verified, tested, and used for seismic 3D soil-structure interaction models up to 25,000 nodes including up to 5,000 interaction nodes. However, for 20,000 node or slightly larger-size SSI problems, the standard solver becomes numerically inefficient on typical PCs with 16GB RAM, since the SSI analysis runtime and the disk storage go up out of hand.

The ACS SASSI Version 3.0 *fast-solver* code, called Option FS, has been extensively verified, tested, and used for coherent seismic 3D SSI models up to 100,000 nodes including up to 35,000 interaction nodes. The *fast-solver* code is much more numerically efficient than the *standard solver* code. The ACS SASSI Version 3.0 *fast-solver* code has two major SSI problem size limitations for current MS Windows PC platforms: 1) MS Windows OS limitation: The maximum accessed RAM for the SSI problem is limited to 192 GB RAM for Windows 7 and 512 GB RAM for Windows 8, respectively, and 2) ACS SASSI limitation: The total node number should be less than 100,000. The governing limitation of the SSI problem size is due to the MS Windows OS limitation. On MS Windows PCs with 16 GB RAM, SSI problems with sizes up to 100,000 nodes including up to 8,000 interaction nodes can be run efficiently with the fast-solver using the in-core SSI solution algoritm. For the SSI problems including larger-size models with more than 80,000 nodes and 8,000-25,000 interaction nodes, MS Windows PCs with RAM ranging from 32 GB up to 192 GB are recommended. For large-size SSI problems with more than 20,000-25,000 interaction nodes, MS Windows 8 PCs with up to 512 GB RAM are recommended.

Table of Contents

1. INTRODUCTION

The ACS SASSI-ANSYS interfacing capability provides an advanced two-step SSI approach that is capable of including more refined FEA structural models, local nonlinear material, and/or nonlinear geometric aspects within the structure or at foundation interface with the soil. There are two ACS SASSI-ANSYS interfacing options: i) *Option A* or ANSYS, and ii) *Option AA* or Advanced ANSYS. Three demo problems, Demo 5, 6 and 7, are provided to help users understand how to best use the ACS SASSI-ANSYS interface via Options A and AA.

The *Option A* or *Option ANSYS* of the ACS SASSI-ANSYS interfacing capability is based on an integrated two-step SSI approach in which the 1st step is the overall SSI or SSSI analysis using ACS SASSI and the 2nd step is the detailed structural stress analysis using ANSYS with the input boundary conditions defined by the SSI responses computed with ACS SASSI. The LOADGEN module (that is a part of the ACS SASSI MAIN module GUI) is used to transfer the data from the ACS SASSI results database to the ANSYS input files.

Option A works with both the standard solver and the fast-solver implementations.

The *Option AA* or *Option Advanced ANSYS* of the ACS SASSI-ANSYS integration capability consists of using directly an ANSYS structural model for SSI analysis without the need for converting the structural model to ACS SASSI. The ANSYS structural stiffness, mass and damping matrices are used directly by ACS SASSI for SSI analysis. The SSI relative displacements, absolute accelerations and response spectra computed for the ANSYS structural FEA model are obtained using the ACS SASSI software. The *Option A* should be used to transfer the SSI response motions at all or selected critical steps as input boundary conditions for the ANSYS superstructure model only for computing structural stresses.

Option AA works with the fast-solver implementation only.

2. USING THE SUBMODELER MODULE FOR ACS SASSI AND ANSYS FE MODELING

The SUBMODELER module has similar functionalities to the PREP module, except the graphical capabilities which are not in SUBMODELER. In addition to the PREP commands, SUBMODELER has many new commands to check FE modeling and generate both the ANSYS and ACS SASSI models, and also convert them back and forth, as needed by the user.

NOTE: *The fact that SUBMODELER duplicates some of the PREP functionalities is due to our intention to replace the PREP module by the SUBMODELER module in the future. The new coming PREP based on the SUBMODELER module will have many additional pre- and post-processing capabilities than the present PREP module.*

SUBMODELER can use all the PREP commands used for building SSI models or selecting SSI analysis options, including the AFWRITE command. The SUBMODELER also includes new powerful commands for handling and combining multiple ACS SASSI or ANSYS models as described in this section. The SUBMODELER is deficient in comparison with the PREP module only on the graphical processing aspects, since it has no graphical capabilities at this time.

SUBMODELER can be launched by selecting *RUN→ "ANSYS Soil Model Generator"* from the ACS SASSI MAIN menu.

As shown in the next sections, SUBMODELER is used in *Option A* to create the surrounding soil deposit ANSYS model, or convert ACS SASSI models to ANSYS or vice-versa, and in *Option AA* to transfer ANSYS model information to ACS SASSI to perform the SSI analysis using the ANSYS structural model directly with no need for conversion to ACS SASSI.

SUBMODELER has also a set of commands that permit to the user to compute the Section-cut forces and moments. For example, the user can create plane section-cuts for selected submodels, such as a concrete shearwall, and then, compute the cross-sectional forces and moments for different section-cuts at different floor levels. For details, please see Demo 8 and Problem 47 in the Verification Manual.

2.1. SUBMODELER Converters for ANSYS Models

The SUBMODELER also has a powerful capability to convert models from ANSYS (.cdb file) to ACS SASSI models. The SUBMODELER Converter has much fewer limitations than the MAIN Converter. The ANSYS model conversion limitations for MAIN Converter are described in the MAIN User Manual. In this section, only the SUBMODELER Converter limitations are described.

The SUBMODELER Converter is capable of converting ANSYS Version 13-14 models that might have element types that are not fully compatible with the ACS SASSI element types. The SUMBMODELER Converter can handle the following element types: i) SOLID element types; SOLID45 and SOLID185, ii) SHELL element types; SHELL63 and SHELL181, iv) BEAM element types; BEAM44 and BEAM188, v) PIPE element types; PIPE188 vi) COMBIN element types; COMBIN14, vii) Couple nodes (CP command), viii) Constraint equations (CE commands), ix) Multipoint constraint element types; MPC184: Rigid Link and/or Rigid Beam, and x) FLUID element types: FLUID80.

For ANSYS models with elements that are compatible with ACS SASSI, such as BEAM4, BEAM44, SOLID45, SHELL63, COMBIN14 and MASS21, the SUBMODELER Converter has the same limitations as the MAIN Converter with the exception that it can handle inputs from the ANSYS Versions 13-14 models, and therefore, it is not limited to ANSYS Versions 11-12 as the MAIN Converter is.

The SUBMODELER Converter limitations are described below.

The Material Data must be defined using the MPDATA command unless otherwise specified. Any other way of defining this data is not recognized by the Converter.

The BEAM4/BEAM44 elements are convertible with some limitations: These beams must be defined with the K node definition. These beams must have properties defined by the RBLOCK, and the RBLOCK entry for the property must have 6, 8, 10, 12 or 19-24 fields for this element to be converted properly. The end releases are defined by KEYOPT 6 and 7. The cross-section commands are limited to SECTYPE, SECBLOCK and SECDATA.

The BEAM188 and PIPE288 elements are convertible with the following limitations. Pipes are converted to equivalent beams only for visual simulation. These elements must be defined with a K node definition.

WARNING: *End releases for these elements cannot be defined for these beams because ANSYS ENDRELEASE command creates new nodes and couples these nodes to simulate an end release. This cannot be properly represented in ACS SASSI. The beam properties must be defined by using the section commands. Beams must be defined with the ASEC or RECT type option to be convertible. Section Offset must be default or not specified for beams (CENT is default for beam) and must not be specified for pipes.*

The ANSYS FLUID80 elements are convertible to equivalent solid elements for visual simulations. Please note that the FLUID80 element could be used to model fluids contained within pools and vessels having no flow rate. Per ANSYS documentation, this linearized fluid element is particularly well suited for calculating fluid pressures and fluid-structure interaction effects for both static and dynamic applications.

WARNING: *The ANSYS FLUID80 elements should be used together with the COMBIN14 spring elements to connect the fluid elements to the structural walls. The CP commands should not be used in conjunction with the FLUID80 elements. For more application details, please see the Problem 48 in the Verification Manual.*

WARNING: *The ACS SASSI Option AA analysis using the ANSYS FLUID80 elements is based on an approximate linearized approach for the fluid modeling in flexible pools and tanks that is not theoretically exact, but is definitely more accurate than the simple mass-spring SDOF model approaches which have been used in practice over the last few decades.*

The COMBIN14 spring element with selected options is equivalent with the ACS SASSI SPRING element type. COMBIN14 is convertible with the following limitations: KEYOPT 2 or KEYOPT 3 must be defined for the group. KEYOPT 2 will take precedence if both KEYOPT 2 and KEYOPT 3 are defined. The spring constant must be defined by the by RBLOCK entry. KEYOPT 2 options 1-6 are supported and options 1-2 for KEYOPT 3.

The MASS21 elements are equivalent with nodal masses defined in ACS SASSI. Mass data must be defined in the RBLOCK entry.

The SHELL63 is equivalent to the ACS SASSI SHELL element type. The SHELL thickness must be defined by RBLOCK entry. The SHELL181 is also convertible for visual simulation only with the limitation that the shell thickness must be defined using section commands. The cross-section commands are limited to SECTYPE, SECBLOCK and SECDATA.

The SOLID45 is equivalent to the ACS SASSI SOLID element type. The SOLID185 is also convertible for visual simulation with the limitation that KEYOPT4 which sets the option for nonuniform materials is not defined or set to 0.

The SUBMODELER menu also includes the “Export to ANSYS” menu option that has an identical functionality with the ANSYS command. The active model in the SUBMODELER is exported in the ANSYS Versions 11-12 APDL format when this option is used.

WARNING: *The back conversion from ACS SASSI to ANSYS is problematic for BEAM44 when the BEAM end releases are different from beam element to beam element inside of the group of the BEAM elements. This is because ANSYS does not accept variations in beam releases from element to element for the same ETYPE command that is equivalent to ACS SASSI Group commands. In this case the structure of the ACS SASSI model and ANSYS model are not compatible. For these situations we suggest use of the ANSYSREFORMAT command that regroups all the beam elements into groups that have a common set of end releases and creates a new ACS SASSI model that is compatible with ANSYS model.*

WARNING: *The SUBMODELER Converter will convert the ANSYS model in ACS SASSI format so that it can be displayed in the PREP, but this model will be flagged by SUBMODELER and the .hou file that the AFWRITE command generates will not be runnable using the standard HOUSE module. If the SSI model isn't runnable with standard HOUSE module, the user will receive a warning when the file is converted. The list of elements types below are the additional element types that can be converted in the ACS SASSI PREP format for graphical processing. There are no limitations on these elements because only element node connections need to be translated for display*.

The list below specifies the element type from ANSYS and the element type used to represent it in the ACS SASSI model:

- TRUSS180 will be displayed as a Beam
- MPC184 will be displayed as a Spring
- PIPE188 will be displayed as a Beam
- FLUID80 will be displayed as a Solid

Other elements will be ignored by the SUBMODELER Converter and not be included in the model.

To run the SSI analysis for these converted ANSYS models that have different element types than the ACS SASSI element types, a new, modified HOUSE module called HOUSEFSA is required. This HOUSEFSA module is a part of the Option AA capability that is described in Section 4.

2.2. SUBMODELER Commands for Checking and Building Complex SSI FEA Models

In addition to the PREP commands for building FE models, SUBMODELER has many additional commands for checking and building FE models which are described here, as follows:

For reading and writing SSI model files:

INPUT, <filename> - This command provides the same functionality as the menu path File->Input used for loading an input text file. The input file name should include the full path, unless the model name and path have been specified using the MDL command.

ACTM, <N> - The ACTM command switches the active model to the Model number N. The initial Model number when SUBMODELER is started is 0. N can be any integer number.

MDL, <filename>, <path> - Create the path for the active Model. The path and model name will also be used to define the path and file name used by the WRITE and ANSYS commands

DMODEL, <N> - Delete the FE model number N.

ANSYS, <filename>, <path> - The ANSYS command writes the model in an ANSYS APDL input file (extension .inp). **ANSYS, <filename>,<path>** must be used if the active model name and path have not yet been specified using the MDL command. If the model name and path have been defined, then no arguments are necessary, and the ADPL file will be saved to the active model directory

WRITE, <filename>, <path> - The WRITE command writes the model in an ACS SASSI PREP input file (extension .pre). **WRITE,<filename>,<path>** must be used if the active model name and path have not yet been specified using the MDL command. If the model name and path have been defined, then no arguments are necessary, and the ADPL file will be saved to the active model directory

For building SSI models:

ANSYSREFORMAT, <Org>,<Map> creates an ACS SASSI model input that has beam end releases compatible with ANSYS model input structure. This command takes an ACS SASSI model and regroups the beam elements based on their set of end releases, so that the beam end

releases can be translated into an ANSYS ADPL format correctly. This command should be used with an empty active model. The user specifies which model should to be converted.
< Org> – Model number of the model to be reformatted
<Map> - A mapping file that indicates the correspondence between the original and the new beam groups

SOILMESH, <Model>,<Scale X>,<Scale Y>, <Hori>, <Vert>,<mX>, <mY>, <Thick>,<Contact> ,<RC num> - The SOILMESH command creates a soil FE mesh for the active model and stores the model data in the user specified Model.

<Model>- User specified integer model number.
<Scale X> - Percentage of growth in the X direction of each level, i.e. 0.07
<Scale Y> - Percentage of growth in the Y direction of each level, i.e. 0.07
<Hori> - Number of horizontal levels to build away from the embedment.
<Vert> - Number of vertical levels to build away from the embedment.
<mX> - Centroid correction in the X direction
<mY> - Centroid correction in the Y direction
<Thick> - Thickness of each new level.
<Contact> - If equal 0 do not use contact surfaces, else include contact surfaces in the model.
<RC num> - Defines the constant set number to be used in ANSYS for the contact surface Real Constants

WARNING: *If the excavation volume has lateral walls that are not vertical, or if the node layers are different at different excavation levels, then SOILMESH will not work correctly.*

EXCAV,<model>,[delta] - This command creates an excavation volume model for a SSI model that doesn't have an excavation volume. The command will use the lowest vertical z-level grid as a template to create a homogenous mesh model up to ground surface. The new excavation model will be stored in the model ID number given by the user. The ground surface must be defined properly in the model used in the generation. The delta parameter is a factor used to match slight variations in z-level used in some models for the same embeddment level. Models that don't have uniform z coordinates across the floor should use a delta > 0, so that the command doesn't generate multiple levels for the small variations of Z.
<model> – Model ID number to store the new excavation volume
[delta] - allowable distance of z-level variation on a single level (Default = 0). Parameter delta should be entered as a positive floating point number or the default will be used.

WARNING: *If there are outcropping beneath ground surface that do not extend to the bottom z-level the code will not generate excavation volume for these areas.*

MERGESOIL, <Struct>,<Soil>,[Mode],[StiffStiff],[StiffSoft],[SepLevel],[Mapping] – This command is used to merge the structural and the excavation volume models together in a new active SSI FEA model.

<Struct> - Model Number of the Structure
<Soil> - Model Number of the Excavation volume
[Mode] - Merging nodes on the structure excavation interface
= 0 Unbonded lateral foundation-soil interface with side solid
= 1 Bonded lateral foundation-soil interface (default)
= 2 Bonded foundation-soil interface using duplicate nodes connected by stiff springs
= 3 Unbonded foundation soil-interface using duplicate nodes connected by soft springs
[StiffStiff] - Stiff spring stiffness for Modes 2 and 4. (Default = 10^7)
[StiffSoft] - Soft spring stiffness for Modes 3. (Default = 10)
[SepLevel] - Global z-coordinate level for the depth where soil separation occurs
[Mapping] - This is mapping filename for the duplicate node merging

GROUNDELEV,<elev> - This command sets the ground elevation constant for the SSI model. This is constant is also set by the HOUSE command.
<elev> - The ground elevation for the model.

GRAVITY,<grav> - This command sets the gravity constant for the model. This is constant is also set by the HOUSE command.
<grav> - The gravity constant for the model.

NCOM - This command compresses the node list so there are no gaps in the defined node numbering in the current model. The command also updates the node element connections in the model to reflect the new node numbering in the model. The command will maintain relative order of the nodes in the model.

GCOM - The GCOM command will compress group number so there are no gaps in the group numbering. The new group numbering will start a 1. The relative order of the groups will not change and this command does nothing to compress the element numbers in each group (use ECOMPR to element numbers inside of groups).

RMVUNUSED – The RMVUNUSED Command checks elements and interaction nodes in the current model to see which nodes are being used. All unused, non interaction nodes will be removed from the model. This command does not compress node numbers or change element node connections. The command should be used in conjunction with NCOM to compress the node list of a complete model.

INTGEN,<type>,[skip] - automatically generates interaction nodes for different substructuring approaches FV, FI-FSIN (SM), FI-EVBN (MSM), Surface model and Fast FV . The excavation volume must be explicitly defined by the ETYPE command for options 1-3. If the ETYPE of the elements is left to the default values, this command will not work. The <type> argument is the type of iteration node generation to be used:

= 1 Embedded Foundation - Flexible Volume (FV)

= 2 Embedded Foundation - Flexible Interface with Excavation Volume Boundary Nodes, denoted FI-EVBN or Modified Subtraction Method (MSM)
= 3 Embedded Foundation - Flexible Interface with Foundation-Soil Interface Nodes, denoted FI-FSIN or Subtraction Method (SM)
= 4 Surface Foundation (interaction nodes are only at the ground surface level)
= 5 Embedded Foundation - Fast FV including multiple layers of internal interaction nodes

The [skip] argument is only necessary when using type 5. Type 5 gives the option to include intermediate layers of interaction nodes between the surface and bottom of the foundation. The [skip] argument specifies the number of node layers to be skipped before generating the next layer of interaction nodes. For example, setting the [skip] argument to 1 will result in interaction nodes being generated at every other level of nodes in the excavation volume.

ETYPEGEN, <type> - assigns the type of SOLID elements. These SOLID elements can be either a part of the structure or a part of the excavation volume based on ETYPE. The type is implicitly defined by default (ETYPE = 0) when an element is added. Then, during AFWRITE rules are used to determine if implicitly defined elements are structural or excavation *.hou file. However, when using WRITE file the implicit ETYPE definition is maintained in the *.pre file. Some INTGEN options require explicitly defined types of elements to work correctly. Use **ETYPEGEN,<type>** to either explicitly type elements using the AFWRITE rules, or changes the type of all the elements in the model. Depending on the assigned value, the type parameter has the following functionalities:

= 0 Changes all implicitly defined element type to explicitly defined element type
= 1 Sets all elements to structural
= 2 Sets all elements to soil

For checking SSI models:

FIXEDINT checks SSI model to find if there is any fixed interaction node (that is a SSI modeling error). No parameter is needed.

HINGED checks model to find all hinged connections between solids and shell and beams and beams and shells. These hinged connections could indicate potentially incorrect FE modeling, since the node rotations from beams and shells are not transmitted to solids at the common nodes, and the node rotations from beams are not transmitted the in-plane shell rotations at the common nodes (the drilling dof equations have no stiffness terms by default).

KINT - The command will check the K-nodes of all Beams to see if they are also defined as interaction nodes. K nodes that are interaction nodes may cause incorrect simulation results. This command will report the nodes that have this issue to the user and remove the node from interaction node list.

FREESPRING - This command will find all unconstrained node dofs that are only connected to a spring, and warn the user about this condition at the nodes where it occurs.

For improving SSI models:

FIXROT, <Stiff> - This command automatically fixes the unnecessary rotational degrees of freedom and adds soft rotational springs to improve numerical conditioning for shell models (for the Kirckhoff plate element the drilling degree of freedom has no stiffness associated with it, and therefore could produce poorly conditioned or unstable numerical models). Using **FIXROT,<Stiff>** will fix all rotational dofs for nodes that belong only to solid elements or shell elements that are parallel to global system planes and have no connection that provides stiffness for the shell in-plane rotations. The FIXROT command fixes rotations by using the D command to fix rotations described above.

For the shell elements that are not parallel to a global coordinate system planes, the FIXROT command automatically adds low-stiffness in-plane rotational springs to each node of the shells (rotations around the normal to the shell planes). The user can control the rotational spring stiffness using the <stiff> parameter of the command. By default the rotational spring stiffness is 10. The user can change the <stiff> parameter values in the updated .pre file as needed if more or less local stiffness is desired in particular sections of the model. To do this, the user should edit the values of the "SC" commands in the .pre file. The stiff parameter = 10 is an appropriate value for typical nuclear structure models that consist of concrete shells for which the shell bending stiffnesses are at least several thousand times larger if defined in kips-ft/rad. The recommendation based on FE theory is that the stiff parameter should be less than 10% of the shell element bending stiffness. The user will need to use the WRITE command to save all the FIXROT command fixes in an updated .pre input file that then can be reviewed by the user.

FIXSLDROT - This command fixes the rotational degrees of freedom of all the nodes that are only connected to Solid elements.

FIXSPRROT - This command fixes the rotational degrees of freedom of all the nodes that are only connected to Springs and Springs/Solid node connections. If the node is only connected to a Spring the unconstrained degrees of freedom are determined by the spring stiffness. If the node is connected to Springs and Solids the rotational degrees of freedom are determined by the Springs rotational stiffness.

FIXSHLROT, <Stiff> - This command applies rotational soft springs to all nodes that are only connected to coplanar shells. The overall spring stiffness is determined by the stiffness argument of the command and applied along the normal shell's plane.

<Stiff> – Stiffness of the soft springs added to shells to remove the shell drilling rotation singularities (default stiffness value is 10)

WARNING: *The above FIXROT and FIXSHLROT commands are HIGHLY RECOMMENDED to be used for ACS SASSI FE shell models that have shells that are oblique with respect to the*

global coordinate system planes. These oblique shell elements can produce numerically unstable SSI models. The use of FIXROT/FIXSHLROT ensures that no numerical singularities could be produced by the shell drilling equations. The use of FIXROT/FIXSHLROT is highly recommended especially if large-size SSI models are run with the fast-solver SSI modules. Benchmark results obtained against ANSYS for various fixed-base models have indicated that the use of the FIXROT/FIXSHLROT commands is highly beneficial for improving the numerical condition of the FEA shell models. The use of FIXROT/FIXSHLROT makes the two FEA codes provide same results for the identical configuration shell models (using SHELL63 in ANSYS to be consistent with the ACS SASSI Kirckhoff thin plate element formulation). The use of FIXROT/FIXSHLROT has no improving effects when the ANSYS FE shell models are used directly via the Option AA capability.

For the Section-Cut option:

SECDATAOPT, <flag> - This command sets the .ess extension files output request in the STRESS input file. When the flag is set to 1, then, the element center stress time history files, the .ess files, will be saved. Their names will be the same names like the nodal stress .ths files, except that the extension is .ess instead of .ths.

<flag> - User output request for saving the element center stress files with the .ess extension
= 0 - no .ess files will be saved
= 1 - the .ess files will be saved for the entire time history duration

CALCM -This command calculates the total mass of the elements in the active model based on material density and the element volumes, and the total lumped mass based on the nodal masses

CALCCOG - This command calculates the center of mass for the active model

TRANELEM, <model>,<group>,<estart>,<efini>,<incr>

Transfers a list of elements from the active model to another new model. These elements will be written/overwritten in the destination model

<model> – Number of the destination model
<group> – Group number for element transfer
<estart> – Beginning element in the transfer list
<efini> – End number in the transfer list
<incr> – Element number increment

TRANVOL,<model>,[Xmin],[Xmax],[Ymin],[Ymax],[Zmin],[Zmax]
Transfer all of the elements in a user defined volume into another model specified by destination reference number. The volume is cube specified by the minimum and maximum in global coordinates. The user can leave any of the boundary arguments blank this will make the code use the default of the model extent for that argument.
<model> – Number of the destination model
<Xmin> - Minimum x of the box

<Xmax> - Maximum x of the box
<Ymin> - Minimum y of the box
<Ymax> - Maximum y of the box
<Zmin> - Minimum z of the box
<Zmax> - Maximum z of the box

CALCMOI,<normalX>,<normalY>,<normalZ>,<positiveX>,<positiveY>,<positiveZ>, <sysno>

This command calculates the inertia moment of a given (cut) area for the current active model in a local coordinate system to be defined by the parameters of this command. The command is useful for any plane that could be eventually a section cut plane
<normalX> – x component of the normal vector of the selected (cut) plane in global coordinates
<normalY>– y component of the normal vector of the selected (cut) plane in global coordinates
<normalZ> – z component of the normal vector of the selected (cut) plane in global coordinates
<positiveX> – x component of the local system orientation for X axis direction in global coordinates
<positiveY> – y component of the local system orientation for X axis direction in global coordinates
<positiveZ> – z component of the local system orientation for X axis direction in global coordinates
<sysno> – the local system number where the local system will be stored
For a horizontal plane in which a local system in the plane with local X in the direction of global X, the input parameters should be 0,0,1,1,0,0 with the default 1 for the system number.

CalcPar, <normalX>,<normalY>,<normalZ>,<positiveX>,<positiveY>,<positiveZ>,<sysno>

This command same parameters as CALCMOI. It computes for a given (cut) area centroid, the moment of inertia and current stress calculation for the active model.
<normalX> – x component of the normal vector of the selected (cut) plane in global coordinates
<normalY>– y component of the normal vector of the selected (cut) plane in global coordinates
<normalZ> – z component of the normal vector of the selected (cut) plane in global coordinates
<positiveX> – x component of the local system orientation for X axis direction in global coordinates
<positiveY> – y component of the local system orientation for X axis direction in global coordinates
<positiveZ> – z component of the local system orientation for X axis direction in global coordinates
<sysno> – the local system number where the local system will be stored

CutVol, <cutnum>,[Xmin],[Xmax],[Ymin],[Ymax],[Zmin],[Zmax]

This command adds new elements to the section-cut data defined by the user with the <cutnum> argument by using volume selection. The selected volume is defined by the other arguments in the command. The user can leave and or all of the volume boundaries empty. By default any boundary argument left empty will be replaced by the appropriate minimum or maximum for the active model.
<cutnum> – Number of the destination cut
<Xmin> - Minimum x of the box
<Xmax> - Maximum x of the box
<Ymin> - Minimum y of the box

<Ymax> - Maximum y of the box
<Zmin> - Minimum z of the box
<Zmax> - Maximum z of the box

CutAdd,<cut num>,<groupnum>,<elem 1>, … <elem N>
CutAdd,<cut num>,<groupnum>,RANGE,<estart>,[efini], [incr]
Both above versions of this command allow the user to add new elements to a section-cut data defined by the user with the <cutnum> argument. The first version of this command is intended for the user to enter a disjointed list of elements from one group while the second command is intended for use with a continuous list of elements or a list with regular gaps between desired elements. This command will create the section cut in memory if no cut has been defined.
<cutnum> – number of the cur
<groupnum> – number of the group where the elements will be added
<elem 1>.., <elem N> – element numbers to be added
<estart> – first element to be added
<efini> - last element to be added (default is the <estart> value)
<incr> - increment to the next element (default: 1)

SLICE, <cutnum>,<pointX>,<pointY>,<pointZ>,<normalX>,<normalY>,<normalZ>
This command will generate in memory user defined section cut plane <cutnum> with all of the elements that cross the infinite plane. The cut plane is defined by a point in space and its orientation with respect to the global coordinate planes
<cutnum> – section-cut number
<pointX> – x coordinate of the selected point on the plane
<pointY> - y coordinate of the selected point on the plane
<pointZ> - z coordinate of the selected point on the plane
<normalX> – x component of the normal vector of the selected (cut) plane in global coordinates
<normalY>– y component of the normal vector of the selected (cut) plane in global coordinates
<normalZ> – z component of the normal vector of the selected (cut) plane in global coordinates

CutRmv,<cut num>,<groupnum>,<elem 1>, … <elem N>
CutRmv<cut num>,<groupnum>,RANGE,<estart>,[efini], [incr]
The two versions of this command are used to remove elements from a section-cut data structure. The parameters are similar to the CUTADD command.
<cutnum> – number of the cur
<groupnum> – number of the group where the elements will be added
<elem 1>…<elem N> – element numbers to be added
<estart> – first element to be added
<efini> - last element to be added (default is the <estart> value)
<incr> - increment to the next element (default: 1)

CutCLR, <firstcut>, [lastcut],[step]
Clear a cut or list of cuts from memory. Once the cut is cleared it no longer exist in memory. The user can redefine it by elements to the cut again but attempts to use the cut for submodeling

purposes will yield errors.
<firstcut> – number of the first cut to be deleted
<lastcut> – number of the last cut to be deleted
<step> – step between deleted cuts

Cut2Sub,<cutnum>,<dest>,[solid]
This command make a new model or submodel out of the section-cut information defined by the user. The active model and the user defined cut data are used to create the new model/submodel. The user also has the option to create a thick shell model using the solid option. The thick shells should be used ONLY for visualization, because the geometry generation algorithm has no way to test if the newly formed solids intersect each other.
<cutnum> – Number of the cut to be used
<dest> – number of the destination model
<solid> – Turn all shells in to solid elements if solid >= 1 (Default solid = -1)

CSECT, <model>, <cutnum>,<X>,<Y>,<Z>,<normalX>,<normalY>,<normalZ>
The command creates a new section-cut model/submodel based on the section-cut data defined by the user. The user must define the section-cut to be used and the destination model of the cross-section, as defined by the first two arguments. The other arguments are used to define an infinite plane by a point on the plane and the orientation of the normal to the plane. The SUBMODELER will then take a cross-section of all the elements in the cut at the infinite plane. The elements of the cross-section will be expanded to a unit thickness, so that the user can visualize the cross-sectional model. This model can then be used to find the section-cut forces and moments at the center of gravity of the section-cut.
<model> – The section-cut model number
<cutnum> – section-cut number
<pointX> – x coordinate of the selected point on the plane
<pointY> - y coordinate of the selected point on the plane
<pointZ> - z coordinate of the selected point on the plane
<normalX> – x component of the normal vector of the selected (cut) plane in global coordinates
<normalY>– y component of the normal vector of the selected (cut) plane in global coordinates
<normalZ> – z component of the normal vector of the selected (cut) plane in global coordinates

READSTR, <Filename>, [Dir]
This command allows the user to read a single stress frame file to compute the section-cut forces and moments. Once this data is read, the section-cut command to get the section forces and moments is the CALCPAR command.
<Filename> – name of the .ess extension stress file to be loaded onto the file including the path.
<Dir> – directory where the stress frame file is to be found

CalcStressHist,<inpfile>,<cutnum>,<pointX>,<pointY>,<pointZ>,<normalX>,<normalY>, <normalZ>,<positiveX>,<positiveY>,<positiveZ>,<sysno>,<outfile>
Calculates forces and moments of a user defined cross-section based on the element stress history frame files, the .ess extension files. This allows the user to batch the section force and

moment calculation of a section-cut for every time step. The computed section-cut force and moment histories will then be written to the <outfile>. The <inpfile> should have the same format with the PREP animation files, such as the .thani extension file

<inpfile> – full path of the input file that contains the list of the .ess frame files.
<cutnum> – The cut to be used
<pointX> – x coordinate of the selected point on the plane
<pointY> - y coordinate of the selected point on the plane
<pointZ> - z coordinate of the selected point on the plane
<normalX> – x component of the normal vector of the selected (cut) plane in global coordinates
<normalY>– y component of the normal vector of the selected (cut) plane in global coordinates
<normalZ> – z component of the normal vector of the selected (cut) plane in global coordinates
<positiveX> – x component of the local system orientation for X axis direction in global coordinates
<positiveY> – y component of the local system orientation for X axis direction in global coordinates
<positiveZ> – z component of the local system orientation for X axis direction in global coordinates
<sysno> – the local system number where the local system will be stored
<outfile> – full path of the output file for the section-cut force and moment histories.

For the Option AA analysis when ANSYS FLUID80 elements are used:

FILLPOOL, **[Stiff],[Sensitivity],[EmptyLevels],[ShellArea],[Offset],[stiff2]**
This command fills a pool with solid fluid elements including a spring pool water/wall interface and the surface areas of the interface using shells. The pool is filled using an algorithm used in the EXCAV command where the floor of the pool and the wall Z-levels are used to create group solid elements to fill the volume. The interface of the pool wall/water are connected by a set of springs with the stiffness of these springs determined by the user. The user should create a FE model of the pool to be filled before using this command. The pool FE model should only contain the walls and floor of a single pool to be filled. The walls and floor must be made of either shells or solids.
<Stiff> - Stiffness of the water wall spring interface parallel to the normal (Default 10^6)
<Sensitivity> - allowable tolerance variation in Z coordinate on the same Z-level (Default 0)
<EmptyLevels> - Number of Z-levels, starting at the highest level, not to be filled with water (Default is 0, that is pool is entirely filled with water)
<ShellArea> - This parameter should be skipped by leaving blank its field.
<offset> – the user-defined starting node number for numbering of the pool elements. This number should be greater than or equal to last node number from the original FE model if the user intends to import the water and spring group back into the original model. If the offset is less than or equal to 0 the pool wall node number maximum will be used (Default -1) an error will occur for any positive number that is less than the pool wall node number maximum.
<stiff2> - Stiffness of wall spring interface in tangential direction to the wall (Default 0)

3. OPTION "A" OR "ANSYS"

The "Option A" or "Option ANSYS" ACS SASSI-ANSYS interfacing capability is based on an integrated two-step SSI approach in which the 1st step is the overall SSI or SSSI analysis using the ACS SASSI model and the 2nd step is the detailed structural stress analysis using the ANSYS model with the input boundary conditions defined by the SSI responses. The LOADGEN module (that is a part of the ACS SASSI MAIN module GUI) is used to automatically transfer the data from the ACS SASSI result database to the ANSYS input files. The 2nd step using ANSYS has two distinct functionalities:

i) Perform structural stress analysis using refined ANSYS FE structural models with detailed meshes, eventually including enhanced element types, non-linear material and plasticity effects, contact and gap elements, and

ii) Compute seismic soil pressure on basement walls and slabs including soil material plasticity, foundation soil separation and sliding using refined ANSYS FE soil deposit models.

The 1st functionality involves creating a more "detailed" FE model in ANSYS that corresponds to the "coarse" ACS SASSI structural model, while the second functionality implies creating an FE submodel of the surrounding soil deposit in ANSYS. Figures 1 and 2 show the new ANSYS FE models created by the ACS SASSI-ANSYS interfacing tools that correspond to the two functionalities mentioned above. It should be noted that the two-step SSI analysis is based on a "cascaded" analysis assumption in which the SSI responses output from the first step (ACS SASSI SSI analysis) becomes the input boundary conditions for the second step (ANSYS stress analysis). In the second step of ANSYS analysis, local nonlinear material or geometric aspects can be considered. The "cascaded" assumption implies that there is no feedback effect due to the local structural and foundation nonlinearities on the SSI soil motions at the foundation-soil interface. This assumption appears to be reasonable for practical applications, except for some particular situations when the foundation separation from the surrounding soil is quite large.

The Option A two distinct functionalities are handled by three separate software modules, the Converter module, the LOADGEN module and the SUBMODELER module. The LOADGEN and SUBMODELER modules are two standalone modules, while the Converter module is part of the SUBMODELER module.

It should be noted that in addition to the Converter included in the SUBMODELER, there is another standalone Converter module that is included in the MAIN module menu. This older Converter is still available, but has fewer capabilities than the newer Converter included in the SUBMODELER module. The key difference is that the MAIN standalone Converter module is limited to the ANSYS Versions 11 and 12, while the SUBMODELER Converter can translate models from the ANSYS Versions 13 and 14 in addition to the ANSYS Versions 11 and 12.

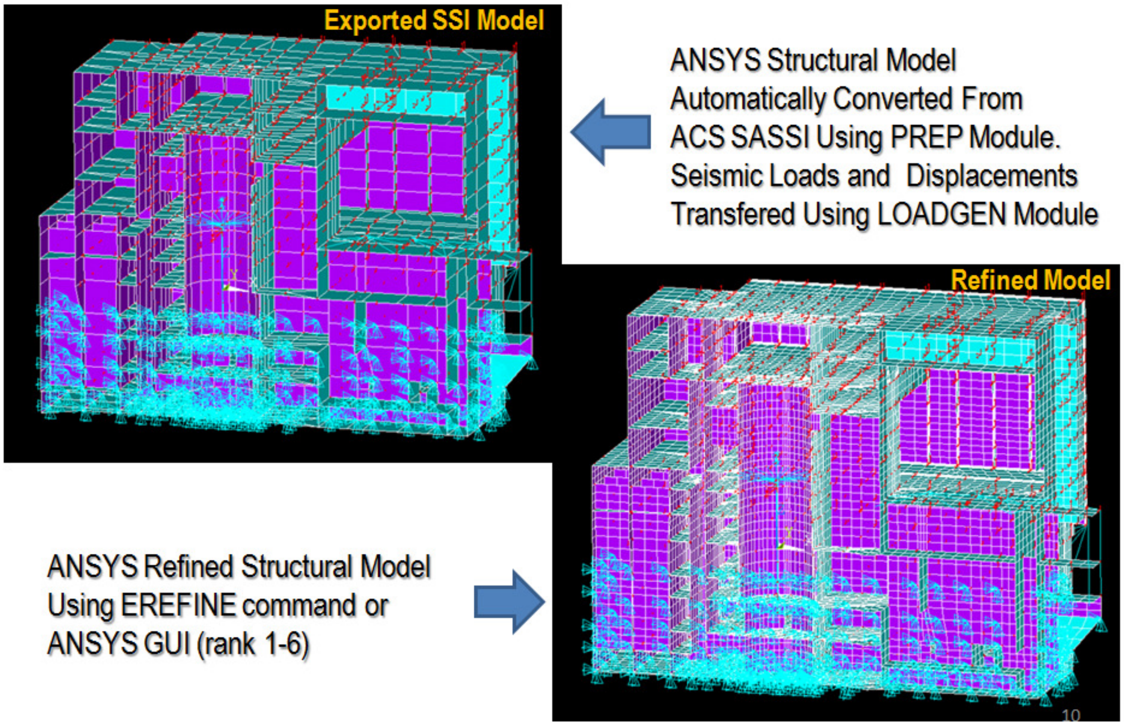

Figure 1 *Coarse* Model (identical to SSI structural model) vs. *Detailed* Model (user defined)

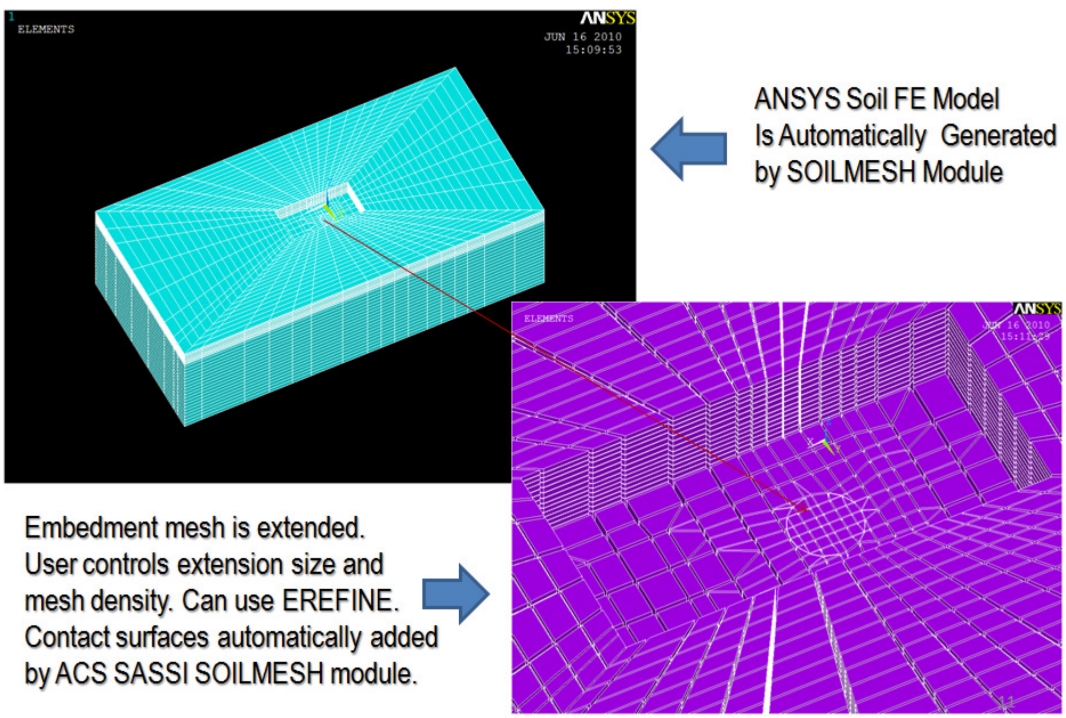

Figure 2 Surrounding Soil ANSYS Model Generated by ACS SASSI SUBMODELER Module

The Converter module translates the ANSYS models saved in the .cdb input format into ACS SASSI structural models in the PREP or SUBMODELER .pre input format, or vice versa translate the ACS SASSI models in the .pre format into ANSYS structural models in the APDL input format. In the latter case, the ACS SASSI excavation volume part of the embedded SSI models is automatically deleted during the conversion to the ANSYS model. The model conversions are done efficiently via the command line in SUBMODELER.

The LOADGEN module uses a simple GUI window dialog to transfer the seismic SSI boundary conditions, the seismic loading, and relative displacements at the foundation soil interface from the ACS SASSI SSI result database (.acc and .thd frames) to the ANSYS structural model via a ANSYS APDL command file. This can be done either for a single time step, or all time steps, or selected critical time steps.

The SUBMODELER module, similar to the LOADGEN module, uses a simple GUI window dialog to generate a new surrounding soil FE submodel in the ANSYS APDL input format, based on the ACS SASSI SSI model embedment geometry. The LOADGEN module is then used to transfer the SSI relative displacements at the interaction nodes defined at the foundation interface with respect to the free-field motion, and the nodal seismic forces for all active dofs at the selected time steps.

3.1. **SSI Modeling Issues**

The second step analysis performed in ANSYS can be either i) a quasi-static or equivalent static stress analysis performed for the selected critical time steps, or ii) a dynamic stress analysis using the direct integration approach

The ANSYS equivalent-static analysis option is applied at all or selected critical time steps, so that the seismic loading phasing and foundation deformations are correctly included. It should be noted that the ACS SASSI-ANSYS equivalent-static stress analyses at selected time steps are more accurate than those computed using the traditional “ZPA-based approach” that computes the seismic equivalent-static forces based structural the ZPA values. However, for the users’ convenience, the traditional ZPA-based approach can be also applied using the ACS SASSI-ANSYS interfacing capability.

For seismic soil pressure SSI analyses, both the LOADGEN and SUBMODELER modules are used, as explained in this manual. Both linear elastic analyses and nonlinear equivalent-static analyses, including nonlinear soil and foundation-soil separation effects can be considered.

The ANSYS dynamic time-domain direct integration analysis uses all time steps. The major benefit of the time-domain dynamic ANSYS analysis is that is greatly reduces the computational requirements of a direct ANSYS SSI analysis approach by eliminating the need of including additional surrounding soil elements in the SSI FE model. Instead, the SSI boundary conditions corresponding to the foundation absolute displacements could be applied at the foundation-soil

interface nodes of the ANSYS. To get the absolute displacements at the support nodes, the computed SSI relative displacements has to be added with the free-field absolute displacements. An alternate for the boundary condition inputs is to load the ANSYS model with the rigid-body kinematic SSI accelerations and relative displacements at the support nodes. This alternate approach works well for the surface foundations under vertically propagating waves for which there is no kinematic SSI effects, so that the kinematic SSI accelerations are identical with the free-field accelerations. The LOADGEN module functionality support both alternate approaches.

It should be noted that ANSYS direct integration approach is limited in practice to the Rayleigh damping matrix option that assumes that structural damping is frequency-dependent. This is a significant modeling limitation, since for the structural and soil hysteretic materials the damping is independent of frequency, so that the Rayleigh damping can be too crude. The use of the frequency-dependent Rayleigh damping could significantly overdamp the low and the high frequency structural responses. Also, ACS SASSI uses a structural mass matrix that is a mixed lumped and consistent mass matrix, while ANSYS uses either a lumped mass matrix or a consistent matrix for the direct time-domain integration approach. Thus, some visible differences between the ACS SASSI dynamic response and the ANSYS dynamic response are expected.

3.2. Validation of the Two-Step Stress SSI Analysis

The high computational accuracy of the implemented ACS SASSI-ANSYS dynamic and equivalent-static SSI stress approaches are demonstrated in the Verification Manual, Problem 32. This problem includes comparisons of the ACS SASSI and ANSYS displacements and stress results for the fixed-base model, the surface SSI model, and the deeply embedded SSI model.

It should be noted that for the quasi-static (or equivalent-static) analysis, the ANSYS structural stresses and the ACS SASSI structural stresses matching is practically perfect for identical structural FEA models, i.e. when the ANSYS model is obtained using the ACS SASSI automatic converters. However, for the ANSYS dynamic SSI analysis option, the ANSYS and ACS SASSI stresses could be significantly different due to the fact that the mass and damping matrices are different, and the numerical solution algorithms for solving equations of motion are different.

WARNING: *If the ANSYS dynamic SSI analysis option is selected, then, the differences between ANSYS and ACS SASSI SSI results due to different mass and damping matrix formulations, and numerical solution methods, as the direct time-integration integration method in ANSYS and the complex frequency convolution method in ACS SASSI, have to be evaluated by the user before the final SSI production runs are started.*

WARNING: *For embedded structures, the user should compute kinematic SSI structural accelerations (for zero mass structure) and then use them to define the translational and rotational rigid-body acceleration fields, as required to define the seismic load forcing function inputs for the ANSYS dynamic analysis. In addition, the foundation relative displacements with respect to the free-field motions shall be computed for all the excavation levels.*

3.3. SSI Methodology for Two-Step Approach

Before using the ACS SASSI-ANSYS integration capability, the user must convert the ACS SASSI structural model to an ANSYS structural model using the ACS SASSI converter. The converter produces an ANSYS model that is identical with the ACS SASSI structural model. This will be referred to in this documentation as the "*Coarse* ANSYS model".

WARNING: *The user has to make sure that this "Coarse ANSYS model" includes no "D" nodal constrains. This is required since this ANSYS model will be used to generate the mass matrix data via modal analysis option.*

The *Coarse* ANSYS model (converted from ACS SASSI) can be used as a starting point for building a *Detailed* ANSYS structural model by using either the EREFINE command for SHELL models or other options in ANSYS.

The user can also develop a *Detailed* ANSYS structural model directly in ANSYS, not by using the converted ACS SASSI model. The *Detailed* ANSYS does not need to have the node or element numbering, or even the geometry configuration of the *Coarse* ANSYS model (that is identical configuration with the SSI model). However, we recommend as a good practice to include all the nodal points from the Coarse ANSYS model in the Detailed ANSYS model. The seismic loads and relative displacements are transferred from ACS SASSI to ANSYS only based on the node coordinate information, and the applied loads and BCs are placed at the closest coordinate from the nodes in the *Detailed* ANSYS model.

The LOADGEN module ensures the automatic transfer of the SSI responses in terms of nodal seismic forces (based on the node accelerations, computed with MOTION) and nodal relative displacements with respect to free-field motion (computed with RELDISP) to the ANSYS model. The seismic SSI boundary conditions (nodal seismic loads and/or nodal relative displacements) from the ACS SASSI structural model are automatically transferred to the refined, *Detailed* ANSYS structural. Foundation displacements can be also transferred to the surrounding soil submodel. The nodal coordinates of the *Coarse* ANSYS model are used to identify the node locations of the seismic loads and relative displacements in the *Detailed* ANSYS model.

For computing the seismic loads on the *Detailed* ANSYS model, user has two options:

i) Use the lumped mass matrix of the *Coarse ANSYS* model, or

ii) Use the reduced mass matrix obtained from the *Detailed ANSYS* model via the ANSYS Guyan reduction for computing the structural seismic forces (nodal masses multiplied by the nodal accelerations).

If the lumped mass matrix option i) is used, then the *Coarse* ANSYS model needs to be generated using the SUBMODELER. If the reduced mass matrix option ii) is used, then, the *Detailed* ANSYS

model will be used directly for computing the reduced mass matrix data or the master mass that corresponds to the node dofs of the *Coarse* ANSYS model The user must make sure the *Detailed* ANSYS model includes no "D" node dof constrains.

3.3.1. ANSYS Equivalent-Static Structural Stress SSI Analyses

Three options for the ANSYS quasi-static or equivalent-static SSI are implemented. These three options depend on the type of the seismic SSI responses used for the second step analysis:

- i) Accelerations for all structural dofs. This still needs “calibrated” soil springs at the foundation support nodes,
- ii) Accelerations for all structural dofs and Displacements for the foundation-soil interface nodes, and
- iii) Displacements for all structural dofs.

The last two of the above equivalent-static approaches are theoretically “exact” for identical structural FE models in ACS SASSI and ANSYS. Figure 3 reviews the capabilities of the three equivalent-static analysis approaches.

These equivalent-static approaches can use SSI seismic loads and/or displacements that are transferred at each time step or at selected critical time steps. These critical time steps, tk, are those for which maximum structural stresses/forces are reached in different parts of the structure at different time steps. These critical time steps can be selected by the user using the ACS SASSI PREP time-history visualization tools, or more efficiently by using the automatic stress “peak” selection option in the ACS SASSI PREP Batch menu.

Using LOADGEN, the seismic load and foundation displacements at all or selected critical time steps will be transferred automatically to ANSYS model in the APDL format in separate load steps. The final stress results should be obtained by the user by enveloping the absolute value results from different load steps.

WARNING: *No output requests are included in the load step files. The output request is entirely at the ANSYS user discretion and responsibility.*

It should be noted that user is also able to select only a portion of the structure as a submodel for further perform the detailed stress analysis. The Displacements and Acceleration options could be combined, so that SSI relative displacements are transferred at the boundary nodes of the submodel and the seismic forces are applied at the submodel interior nodes. In this case, the user has to select all the structural nodes around the selected portion of the structure as interaction nodes, so that LOADGEN will transfer the node relative displacements of these nodes as SSI boundary conditions into the ANSYS APDL file.

It should be noted that the traditional ZPA-based approach that uses the SSI maximum nodal acceleration response values to compute equivalent-static seismic forces could be applied in

conjunction with the above option i). However, the ZPA-based equivalent-static approach is generally overly conservative, especially for non-symmetric structures, but, it can also be locally unconservative, since it loses the phasing of the seismic loads on the structure. The ZPA-based approach requires that the user determines calibrated soil springs that need to be included at the foundation support nodes, or assume a rigid base. The rigid base assumption that was traditionally accepted in the past could be very crude, especially for flexible foundations in soft soils as illustrated in the Problem 32 of the Verification Manual.

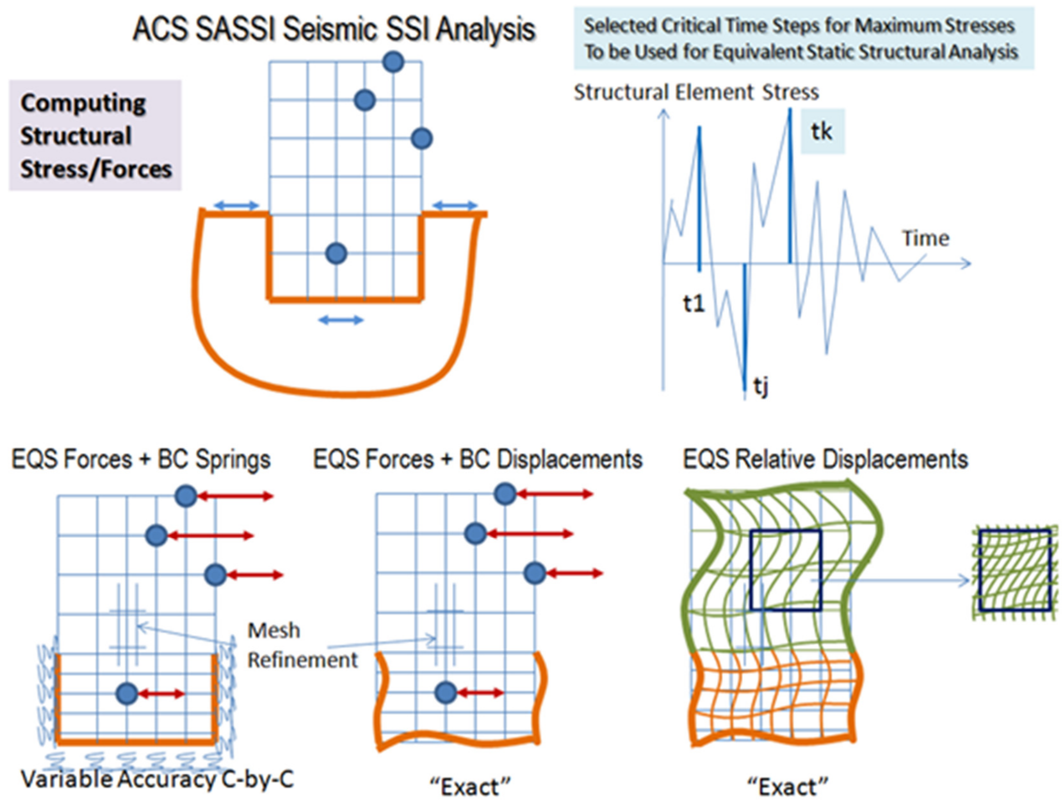

Figure 3 ACS SASSI-ANSYS Equivalent-Static Structural Analysis: Using Acceleration Input (left), Displacement Input (right) and Mixed, Acceleration and Displacement Inputs (middle)

Table 1 describes the different types of equivalent-static stress analysis that were implemented based on the SSI approaches illustrated in Figure 3. Of these approaches, the mixed approach using both the SSI acceleration and displacement boundary condition as inputs shown in the middle of Figure 3 is recommended as the best approach. This approach is particularly useful, when the ANSYS equivalent-static model is much more refined than the SSI model, and/or when improved element types for computing stresses, and/or local structural nonlinearities are included.

Equivalent Static (EQS) Approaches	SSI Structural Response Quantities Used	Boundary Conditions (Loaded DOFs)	Computed Stress Accuracy	Comments
Equivalent Static Inertial Forces (Maximum Values) *Automatic export from ACS SASSI to ANSYS except the soil springs to be computed by the user*	Maximum Accelerations (ZPA)	All Dynamic Structural DOFs. *ANSYS master node masses are computed automatically and then used for inertia force calculations.*	Variable, from reasonable to crude. Seismic load phasing at any given time is lost. The results are globally conservative, but locally could be unconservative, especially for asymmetric structures and flexible foundation structures.	Only one static analysis. Soil spring evaluation is difficult, inconsistent. Affected by the soil spring evaluation accuracy. Some local nonlinearities or local configuration changes could be considered.
Equivalent Static Inertial Forces at t=tk *Automatic export from ACS SASSI to ANSYS except the soil springs to be computed by the user*	Accelerations at t=tk	All Dynamic Structural DOFs. *Same note as above*	Reasonable, especially, if soil springs are well-selected. Include seismic load phasing.	Few to several static analyses. Affected by the soil spring evaluation accuracy. Some local nonlinearities or local configuration changes could be considered.
Equivalent Static Displacements at t=tk *Automatic export from ACS SASSI to ANSYS. User just clicks.*	Displacements at t=tk	All Structural DOFs and Soil Coupling DOFs. *Same note as above*	Theoretically Exact	Few to several static analyses. Could be used for submodeling for selected parts of the structure (subsystems).
Equivalent Static Forces and Displacements at t=tk *Automatic export from ACS SASSI to ANSYS. User just clicks.*	Accelerations and Displacements at t=tk	Dynamic Structural DOFs for Accelerations and Soil Coupling DOFs for displacements	Theoretically Exact	Few to several static analyses. Some local nonlinearities or local configuration changes could be considered.

Table 1 ANSYS Equivalent-Static Seismic Stress Analyses Including SSI Effects

3.3.2. ANSYS Dynamic Structural Stress SSI Analyses

Through the ACS SASSI-ANSYS integration capability, ANSYS can be used to perform an efficient dynamic analysis in the second analysis step using either the *Coarse* or the *Detailed* ANSYS model. In this case, the seismic load can be defined by either i) the kinematic SSI forces on the structure that are introduced by the SSI translational and rotational rigid-body acceleration history of the massless structure, or ii) the absolute displacement histories at all foundation–soil support nodes. The foundation flexibility SSI effects is captured through the displacement histories at the foundation-soil interface nodes.

WARNING: *Due to the differences in the dynamic modeling in ANSYS and ACS SASSI, i.e. different formulations for the mass and damping matrices, and different numerical techniques for solving the differential equations of motion for SSI system, we recommend a preliminary comparative ACS SASSI-ANSYS stress analysis using identical FEA models in the two codes, i.e. the Coarse ANSYS model that is obtained by converting from the ACS SASSI model. Such a preliminary comparison between Coarse ANSYS model results and ACS SASSI results is necessary to validate the ANSYS dynamic model including the Rayleigh damping modeling*

against ACS SASSI results. After the ANSYS model validation is passed, only then, for the final stress analyses, the Detailed ANSYS model can be used instead of the Coarse ANSYS model.

3.3.3. ANSYS Equivalent-Static Seismic Soil Pressure SSI Analyses

Using the ACS SASSI-ANSYS integration Option A capability, the user can efficiently and reasonably accurately compute the seismic soil pressure on the embedded foundation walls and base slabs. Using ANSYS equivalent-static analysis, the seismic soil pressures can be computed for the critical time steps. These critical time steps are usually those that produce the largest base shear, sliding forces and overturning moments.

WARNING: *It should be understood that the soil pressure calculation is an approximate approach that is not theoretically exact for nonlinear analyses.*

The first action taken by the user before performing the seismic soil pressure analysis is to generate the ANSYS submodel for the surrounding soil deposit. This soil model is then used for the soil pressure analysis. Using the SUBMODELER module the user can automatically generate a surrounding soil deposit model full control over the mesh refinement and extension. In general, for an equivalent-static stress analysis, the soil deposit should extend about twice the size of the foundation to eliminate any effect the soil deposit boundaries away from the foundation area may have on the soil pressure results. The user could perform some sensitivity studies on the soil mesh sizes and its extension in lateral and vertical directions to ensure that both are correctly sized for the problem.

NOTE: *The user could use limited extension surrounding soil model with displacement boundary conditions input from SSI analysis.*

Two ACS SASSI-ANSYS equivalent-static approaches are implemented, as shown in Figure 4:

i) *Linear Analysis* (foundation is bonded with the elastic soil) uses as the SSI boundary condition inputs the SSI foundation-soil interface node relative displacements with respect to the free-field motion, and

ii) *Nonlinear Analysis* (foundation can separate from soil material that can behave linear or nonlinear) uses as SSI boundary condition inputs the SSI seismic loads on the structure including basement.

It should be noted that the *Linear Analysis (LA)* option requires only the use of the surrounding soil deposit model with relative displacement boundary conditions. No structural model is needed. The *Nonlinear Analysis (NA)* option requires the use of both of the ANSYS structural and soil deposit models, with the SSI seismic loads transferred to the structure model.

Theoretically, the NA option can be also applied using the ANSYS dynamic time-integration approach, but in this case the model size and the mesh refinement of the soil deposit is

dramatically larger to handle the high-frequency wave components transmission and the reflected waves into the infinite soil media space. Inclusion of a large-size soil model will produce huge computational analysis efforts and therefore, is totally impractical for seismic SSI design-basis analyses.

If dynamic analysis is used, then, the user has to input the desired boundary conditions for the ANSYS soil deposit model. By default, no boundary conditions are placed on the lateral surfaces of the soil deposit model.

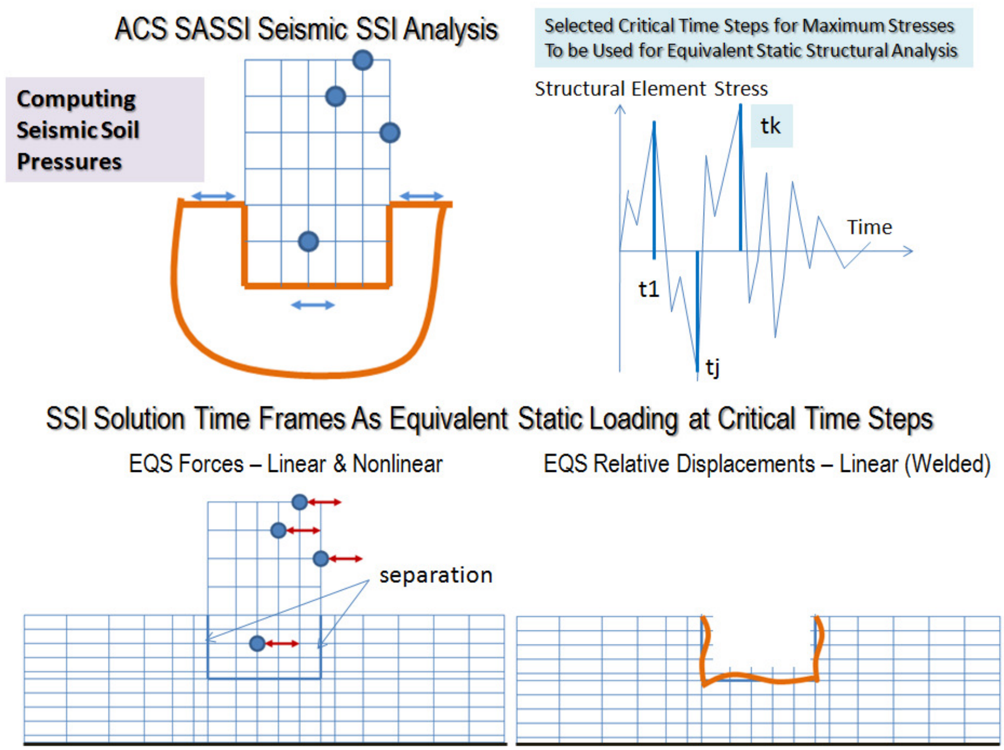

Figure 4 Seismic Soil Pressure Equivalent-Static SSI Analysis: Nonlinear/Linear Analysis Using Seismic Loads (left) and Linear Analysis Using Support Displacements (right)

3.4. ACS SASSI-ANSYS Interface Description and Use

The ACS SASSI-ANSYS interface capability provides an efficient tool to perform the second analysis step using a *Detailed* ANSYS model. The ACS SASSI-ANSYS interface tool (LOADGEN module) automatically generates the SSI boundary condition input files for the *Detailed* ANSYS model. The SSI boundary condition inputs consist of the nodal seismic forces within the structure and/or nodal relative displacements at the foundation interface with surrounding soil. The required SSI boundary conditions are automatically transferred to ANSYS in an APDL format input file. This APDL file contains nodal forces and/or relative displacements for ANSYS equivalent-static analyses, and nodal displacements and acceleration field data for ANSYS dynamic analyses. The

type of SSI boundary condition input is selected by the user from the ACS SASSI SSI analysis result database.

It should be noted that the SSI nodal forces and nodal displacements are transmitted to the ANSYS model based on the structure node coordinates. The nodal seismic load and/or relative displacements with respect to the free-field control motion from the SSI analysis are applied at the same node locations within the Detailed ANSYS model.

The nodal acceleration response data is converted to the nodal seismic forces by multiplying them with nodal masses. The displacement boundary conditions and the seismic forces are generated in the ANSYS APDL format input files, so that the user can apply them directly to ANSYS using the command, "/INPUT, …".

The user has two options for defining the nodal seismic forces for the *Detailed* ANSYS model based on the mass matrix formulations:

1) *Lumped Mass matrix of the Coarse ANSYS model* that is identical to the SSI model (this ANSYS coarse model can be obtained by the user using the new ACS SASSI PREP converter from ACS SASSI to ANSYS), and

2) *Reduced Mass matrix of the Detailed ANSYS model* (using a Guyan reduction of the mass matrix of the Detailed ANSYS model assuming as masters all the active degrees of freedom of the Coarse ANSYS model.

3.4.1. Requirements and Limitations

In this section we describe the use of the ANSYS load generator or LOADGEN module (included in the MAIN GUI menu) to transfer the ACS SASSI SSI analysis result data as input boundary conditions for the ANSYS detailed stress analysis.

Before using the LOADGEN module options the user needs to make sure that the appropriate SSI model and result database are used, since otherwise the LOADGEN module will not function correctly. These must be open in MAIN when LOADGEN is launched. The current database path and model name are shown in the bottom right of the MAIN window.

There are a few things the user needs to pay attention when using LOADGEN:

1) *Avoid having coincident node locations.* Avoid having nodes with the same coordinates. The load generator uses the nodal coordinates from the SSI model to determine which node in the Detailed ANSYS are loaded Therefore, if there are two or more nodes with same coordinates, the ANSYS load generator cannot distinguish between these two nodes. The coincident nodes will cause the displacements or inertial forces to potentially be applied incorrectly. This incorrect load can cause erroneous results in the static analysis. *When there are coincident nodes in the ANSYS refined model, the user should check if the displacement BC's and inertial forces are applied correctly.*

2) *When the "Lumped Mass" option is selected for the "Generate Mass Data" input using the Coarse ANSYS model*, the user must make sure the *Coarse* ANSYS model has no displacement constraints (from the "D" command in ANSYS) at any node. Also, if there are beam elements with end-released degrees of freedom in the *Coarse* ANSYS model, the ANSYS model the nodal masses cannot be extracted, as these are not compatible with the "LUMPM" option for modal analysis. Refer to section 3.4.3 for an explanation on how to deal with this scenario.

3) *When select the "Master Node Mass" option for the "Generate Mass Data" input using the Detailed ANSYS model,* the user has to make sure the *Detailed* ANSYS model has no displacement constraints (from the "D" command in ANSYS) on the model. The reason same as for above item 2).

WARNING: *It is strongly advised to include all the nodes from the Coarse ANSYS model in the Detailed ANSYS model. Since ANSYS will apply the nodal loads to the nodes of the Detailed ANSYS model with the nearest coordinates to the Coarse ANSYS model nodes, it is recommended that all the Coarse ANSYS model nodes to be included in the Detailed ANSYS model to ensure that the loads are not applied to incorrect nodes. Failure to include coarse model nodes with the same coordinates in the detailed model will require the user to extensively check the detailed model to ensure that the loads from coarse model are applied to the correct locations. This checking is a MUST when the detailed model has a different geometry configuration and/or different element types than the coarse model. This applied load checking is labor intensive and has significant associated risks. Also, as above mentioned, the existence of pairs of duplicate nodes with the same coordinates should be avoided in both models since they can generate the same problem of misplacing the nodal loads in the detailed model.*

3.4.2. ACS SASSI-ANSYS Interface Description

The ACS SASSI-ANSYS integration modules, LOADGEN and SUBMODELER (or SOILMESH in previous version), are accessible though the ACS SASSI MAIN module. After using the SUBMODELER model converter to create a *Coarse* ANSYS structural model equivalent to the ACS SASSI structural model, the user will transfer the seismic loads and displacements from the SSI result database to ANSYS model with the LOADGEN module.

The ANSYS load generator (the LOADGEN module) provides four functions to convert the SSI displacements and/or accelerations at all or selected critical time steps from the ACS SASSI results into the ANSYS seismic load via APDL format input file, as shown in Figure 5.

The four options are organized in the first user input data group labeled as "*Data to Add from ACS SASSI to the ANSYS model"*

1. ***"Displacements"*** option provides the function to apply displacements only;

2. ***"Acceleration"*** option provides the function to apply inertial forces only;

3. ***“Displacement and Acceleration*”** option provides the function to apply both displacement boundary conditions and seismic forces from nodal accelerations which here will be referred to as a “mixed” boundary condition;

4. ***“Displacement for Soil model”*** option provides the function to generate the displacement boundary conditions for soil FE model that is used to compute the seismic soil pressures on embedded foundation walls and base slab.

The “*Displacement*” option is the simplest to use, and requires the least amount of input data. The “*Acceleration*” option requires the user to prepare the nodal mass data first. If the “Acceleration” option is selected, then user will need to define soil springs at the foundation support nodes in order to simulate the soil deposit stiffness. The structural seismic forces are automatically calculated using either the *Lumped Mass* matrix option for the *Coarse* ANSYS model, or the *Reduced Mass* matrix option for the *Detailed* ANSYS model. The *Reduced Mass* matrix corresponds to a reduced model produced by the master nodes selected at the locations of all the nodes contained in the *Coarse* ANSYS model. The computation of the soil springs at support nodes will likely not be a trivial calculation for the user, especially for embedded foundations. Alternately, the *“Displacement and Acceleration”* option (or “*Mixed*” option) can be used. This option provides theoretically exact boundary conditions (BCs) for the ANSYS equivalent-static analysis.

We strongly recommend the use of the “*Displacement and Acceleration”* option for application to all SSI problems.

It should be noted that using the *“Displacement and Acceleration”* or *“Acceleration”* options, the user can include local structural nonlinearities in the ANSYS equivalent-static analysis.

The *“Displacement”* option applicability is very limited, since by using it, the user constrains the ANSYS displacement solution to the ACS SASSI displacement solution, even in situations when a much more *detailed* ANSYS model is applied.

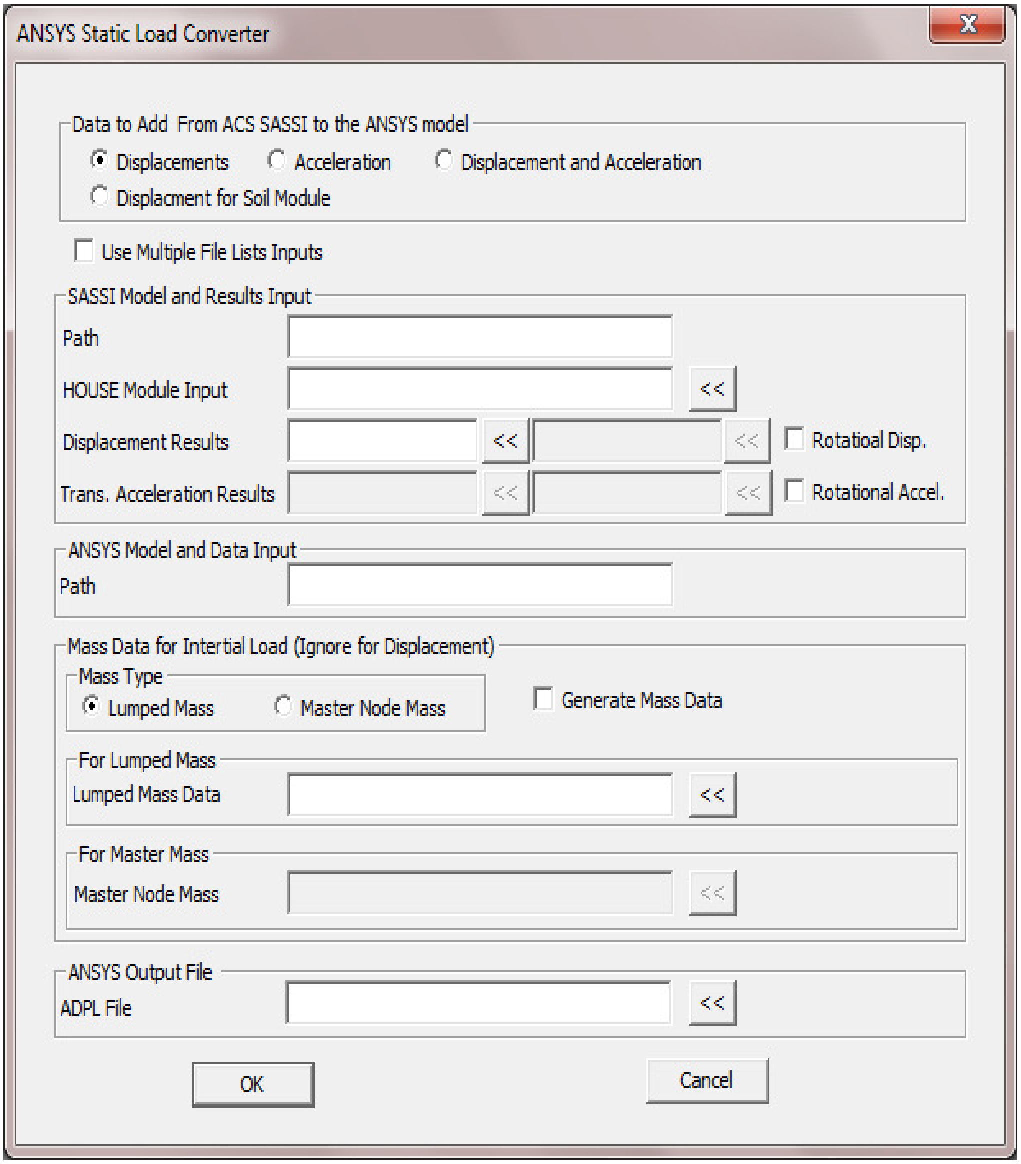

Figure 5: ANSYS Static Load Generator Window Launched from ACS SASSI MAIN

The "*Displacement for Soil Model*" option is designed to generate the displacement boundary conditions for the ANSYS soil deposit model that is generated by SUBMODELER (or SOILMESH in previous revision) module and is used to calculate the seismic soil pressures on foundation walls and mats.

WARNING: *Currently only the Flexible Interface method with interaction nodes at the foundation-soil interface (Subtraction) is supported by default for the "Displacement for Soil Model" option. The application of the Flexible Interface method with additional SSI interaction nodes (Modified Subtraction), or the application of the Flexible Volume method is possible only if the user creates a new SSI model that has the SSI interaction nodes defined only at the foundation-soil interface. Using the AFWRITE command for HOUSE, the user will create a new .hou file that is then copied*

in the ACS SASSI result folder. This .hou file will be identical to the .hou file used for the SSI analysis, with the exception of the interaction nodes.

Seismic load files can be defined either at a single time step or at multiple time steps. In the latter case the user will need to create a file with the list of the SSI response frame file names, rather than use a single file name as input. The format for this file is detailed in the next section.

3.4.3. ANSYS Seismic Load Generator for Exporting ACS SASSI SSI Responses as Input Boundary Conditions to ANSYS Model (LOADGEN Module)

The use of the ANSYS seismic load generator consists of two stages: 1) *Stage 1* is the input file *preparation stage*, and 2) *Stage 2* is the *running stage*, when the LOADGEN module is used to generate the APDL load input files for the ANSYS equivalent-static analysis or dynamic analysis.

To demonstrate how the ANSYS load generator is used, an illustrative example is considered in the next sections, as shown in Figures 6 through 21. All the file paths and names listed below are provided for explanation purposes only. Any other paths or file names can be used, as long as they follow the guidelines listed in this document.

The application of the LOADGEN module for ANSYS equivalent-static structural stress analysis is shown in detail in the Demo 5 problem, which is included on the installation media.

Stage 1: Input File Preparation

Before using the ANSYS seismic load generator, the user must prepare the following data files:

Step 1: Create a folder to save the ACS SASSI model and result files

In this first step, the user creates a folder to save the necessary SSI analysis result files that will be used for the ANSYS equivalent-static analysis. This folder, called "F:\SSI_Results" in this example, should contain the following data files:

1. ACS SASSI HOUSE input file (.hou extension); for illustration purposes, assume that the .hou file is named "solid_box.hou" file.

2. All necessary relative displacement frame data files from SSI analysis;

    a. If an equivalent-static analysis in ANSYS is to be performed, the relative displacement data files at the selected time steps should be copied to this folder. For this example, assume that the names of these data files are "THD_04.215_00844", and "THD_04.105_00822".

b. If a dynamic analysis in ANSYS is to be performed, then all the displacement data files at all time steps should be copied to this folder. For this example, assume that the file names are “THD_00.000_00001”, “THD_00.005_00002”, …, “THD_14.995_03000”.

3. All necessary acceleration frame data files from SSI analysis;

   a. If an equivalent-static analysis in ANSYS is to be performed, the acceleration data files at selected times should be put in this folder, such as “ACC_04.215_00844”, and “ACC_04.105_00822” in the example.

   b. If a dynamic analysis in ANSYS is to be performed, the input ground acceleration history data file, including the six degrees of freedom components in space, should be copied to this folder. In this example, we assume that this file is named “NEWMHX.ACC”.

4. If multiple critical time steps for defining the seismic loads for ANSYS analysis are to be used, then an input file with the list of the frame data files must be created. The first line of this file contains the number of frame files that the user will use to generate seismic load files for ANSYS analysis. The subsequent lines are the list of the file names containing the SSI response frames. To use the multiple file input option, the user should select this option in the LOADGEN GUI. The name of the file containing the list of frame files should be input into the appropriate box for either “Displacement” option or “Acceleration” option. If the *"Displacement and Acceleration"* option is used, then two list files must be created for both acceleration and displacement frames at selected times. Below is an example of a displacement list file, "disp_list.txt". This file will be used to create input BCs for ANSYS analysis using the two selected relative displacement frames.

```
        2
        THD_04.105_00822_E
        THD_04.215_00844_F
```

Step 2: Create a folder to save ANSYS model and input load files

In this second step, the user creates a folder where the ANSYS model and input load files produced from LOADGEN will be saved. This folder, called *“F:\ANSYS_Files*” in this example, should contain the files listed below. Please note that it will be necessary to use two ANSYS models for this step. A copy of the original ANSYS model should be modified to remove all displacement boundary conditions. Any beam releases should be removed as well to ensure compatibility with using the "LUMPM" option. This secondary model will only be used to generate the nodal mass file to be used with LOADGEN. The original model will still be used for the ANSYS analysis. The files contained in the ANSYS working folder described above are as follows:

1. The *Coarse* ANSYS model that is converted from the ACS SASSI SSI model using the *"Export to ANSYS"* menu selection or *"ANSYS"* command in SUBMODELER. The model converter translates the ACS SASSI structural model to an ANSYS APDL file with the extension *".inp"*. This text file is found in the ACS SASSI working folder. The user should copy the file to the ANSYS working folder called "*F:\ANSYS_Files*". The *Coarse* ANSYS model db file is created by using the /INPUT command in ANSYS, and is saved as a .db file. If the user plans to use "Lumped mass" to do the equivalent-static analysis, a *Coarse* ANSYS model for generating lumped mass data should prepared by the following steps:

   1) delete all displacement BCs in the generated *Coarse* ANSYS model;

   2) make sure this model is able to do modal analysis;

   3) save as ANSYS model for generating Lumped mass data.

2. If the user plans to use "Master Mass" for the ANSYS analysis, he should also prepare the *Detailed* ANSYS model for generating mass matrix data for the reduced dynamic model that corresponds to the master nodes defined by all nodes of the *Coarse* ANSYS model by the following step 2.

   1) Copy the *Detailed* ANSYS model in the folder "*F:\ANSYS_Files* ";

   2) Load this model into ANSYS;

   3) Delete all the displacement BCs;

   4) Make sure the model is able to do modal analysis;

   5) Save it for generating mass data at the master nodes.

3. If the user wishes to generate multiple load files in a single run, then the frame name list file should be input in the "APDL File" section box. This file has similar file format as the input file of multiple displacement data file described in Step 1. The first line is the number of load frame files. The subsequent lines are a list of file names that the load data will be saved to. This file name should be entered into the "*APDL File"* box as shown in Figure 7. In the example, this file is called "disp4soil_apdl_list.txt" and its content is as follows:

   2
   disp4soil_822.cmd
   disp4soil_844.cmd

4. For generating the seismic loads for the ANSYS model, the user is required to select the mass matrix generation option that is either *"Lumped Mass"* data option using the *Coarse* ANSYS model, or *"Master Node Mass"* data option using the *Detailed* ANSYS model. Theoretically, the use of the *Detailed* ANSYS model reduced mass data for computing the seismic forces provides a more refined numerical solution than using directly the *Coarse*

ANSYS model lumped mass data for computing the seismic forces. The generation of nodal mass data for computing the seismic forces is described later in this section.

WARNING: *The user should make sure the ANSYS model used for generating mass data is able to do modal analysis, which implies that there are no released node degrees of freedom using "D" command in ANSYS model to be used to calculate the structure mass matrix data.*

Stage 2: Using LOADGEN for ANSYS Equivalent-Static Analysis

After finishing all the preparation work, the user is ready to run the ANSYS load generator. To run the LOADGEN module for equivalent-static analysis, the user needs: first open the database and model from the "Model" menu in the ACS SASSI MAIN; then, select in the ACS SASSI MAIN menu the option *RUN→* "*ANSYS Eq. Static Load*". There are four options can be selected to generate the load files for ANSYS static analysis. The flowchart in Figure 6 shows the basic steps to generate the seismic load files according to the user's selection. In the following sections, an example will be used to illustrate these steps in more details. The paths displayed in the *"ANSYS Static Load Generator*" window are F:\SSI_Results and F:\ANSYS_files for the SSI result files and the ANSYS input and output files, respectively. These directories are the ones the user specified in the Stage 1 as described earlier in this section. Suppose there are the following files in the folder of "F:\SSI_Results".

- "Solid_box.hou" – input for the HOUSE module;
- "THD_04.105_00822", "THD_04.215_00844" – selected displacement frame files at time 4.105 seconds and 4.215 seconds;
- "ACC_04.105_00822", "ACC_04.215_00844" – selected acceleration frame file at time 4.105 seconds and 4.215 seconds;
- "disp_list.txt", "acc_frm_list.txt" – input files that include the names of multiple displacement data files and acceleration data files, respectively. These name list files are required only if "Use Multiple File List Input" box is selected. They are prepared in Stage 1 described earlier in his section.

And there are the following files prepared in the folder of "F:\ANSYS_files".

- "Solid_box.inp"– the ANSYS APDL input file automatically generated by the ACS SASSI PREP converter using the menu selection *"Convert to ANSYS*";
- "solid_box.db"– the Coarse ANSYS model (identical with SSI model geometrically) database produced by ANSYS for lumped mass data option;

- “solid_box_ref.db”– the Detailed ANSYS model database produced by ANSYS for the reduced mass data option using master nodes defined at all nodes of the Coarse ANSYS model (or SSI model) via Guyan reduction;

- “lumped_mass.dat” and “master_mass.dat” – The lumped mass data file and the reduced master node mass data file, respectively, that are produced by ANSYS as explained later in this section

- “disp_apdl_list.txt”, “acc_lump_apdl_list.txt”, and “acc_master_apdl_list.txt”, "mix_lump_apdl_list.txt”,“mix_master_apdl_list.txt”, disp4soil_loads_list.cmd” – The input files that define the APDL output files, when user selects to multiple frame files at different time steps in a single analysis run.

“Displacements” Option (No Nodal Mass Needed)

If the user wishes to apply only the displacements from the ASC SASSI results to the ANSYS model, the “Displacements” option should be used, as shown in Figures 7 and 8.

The user should input the following parameters.

1. Check the *“Use Multiple File Lists Inputs*”, if multiple displacement load files will be generated in one run;

2. Enter the folder name that contains the ACS SASSI results that was prepared in Stage 1, “F:\SSI_Results”.

3. Enter the HOUSE module input file name, which has the “.hou” file extension that was prepared in Stage 1, “Solid_box.hou”.

4. Enter the displacement data file for the selected time, which was prepared in Phase I, “THD_04.105_00822” or “THD_04.105_00844”. If *“Use Multiple File Lists Inputs”* was checked, the file that defines multiple displacement data should be input. In our demonstration it is “disp_list.txt”;

5. If the user wants to apply rotational displacements as well, check the *“Rotational Disp”* check-box first, then input the data file name of rotational displacement in the corresponding edit box. If “*Use Multiple File Lists Inputs*” was checked, the file that defines multiple rotational displacement data should be input in the edit box;

6. Enter the folder name that contains the ANSYS model in the *“Path”* input box that was prepared in Stage 1, “F:\ANSYS_Files”

7. Enter the displacement BC APDL command file name in the text box next to “APDL file”. This file contains the displacement BC described in APDL commands. If *“Use Multiple File Lists*

Inputs" was checked, the file "disp_apdl_list.txt" that contains multiple APDL output file names should be input.

8. Click the "OK" button to run the code and generate the displacement BC file in the ANSYS working folder with the file name entered in Step 7. This file can be used for the ANSYS equivalent-static stress analysis using the APDL command "/INPUT, …"

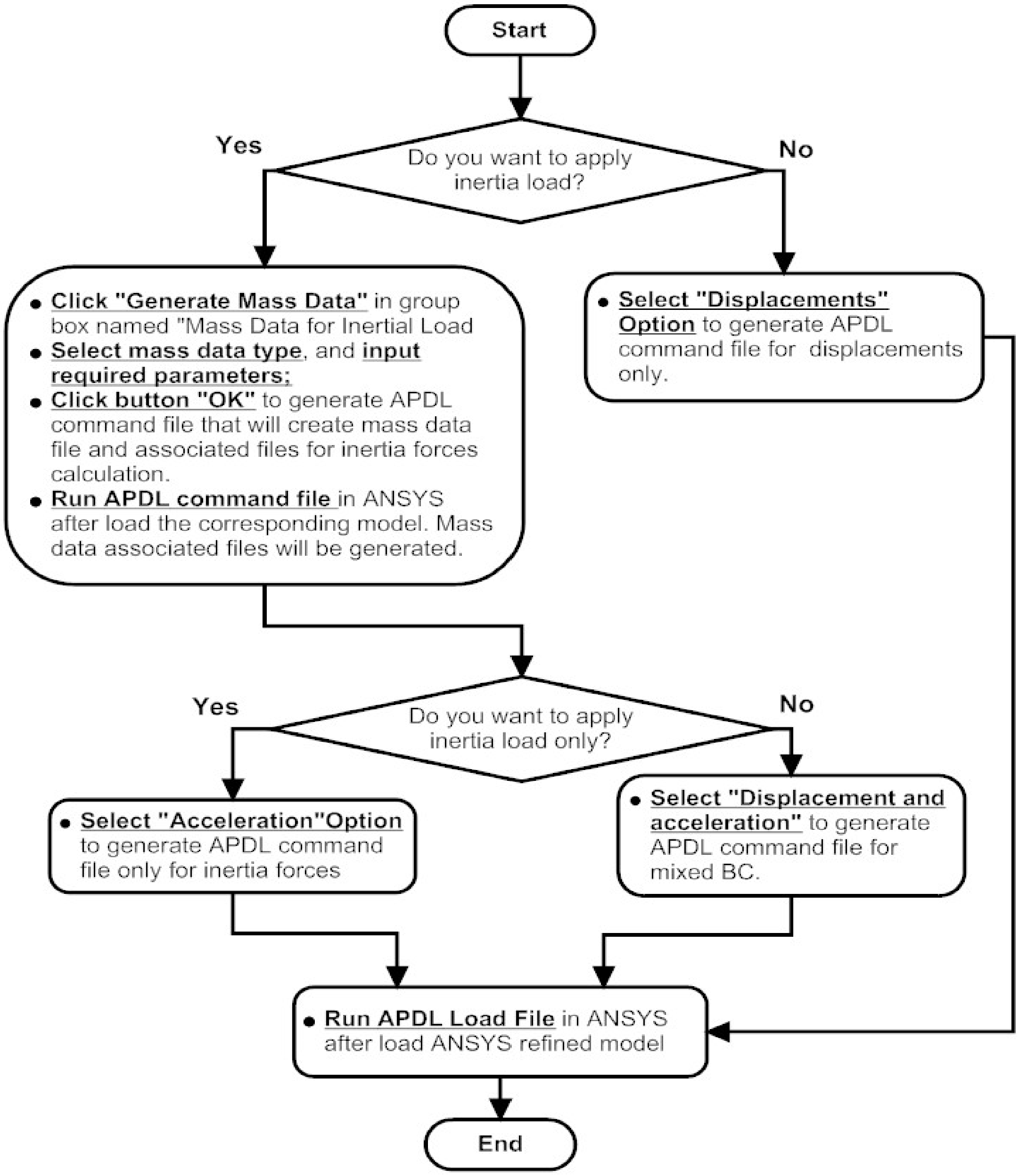

Figure 6: Basic Flowchart for running the ANSYS load generator

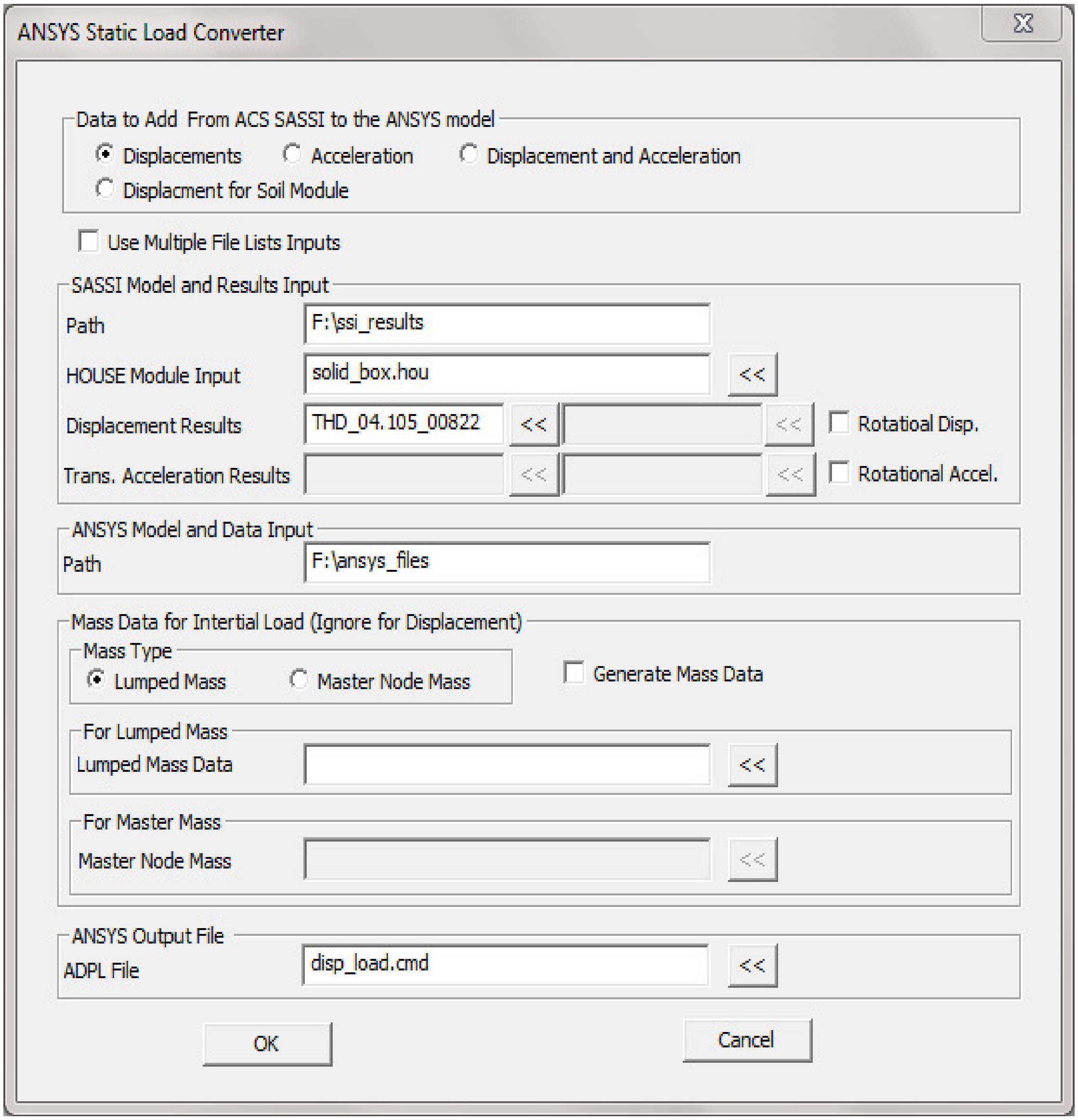

Figure 7: ANSYS Load Generator “Displacements” Option for Single Load Step File

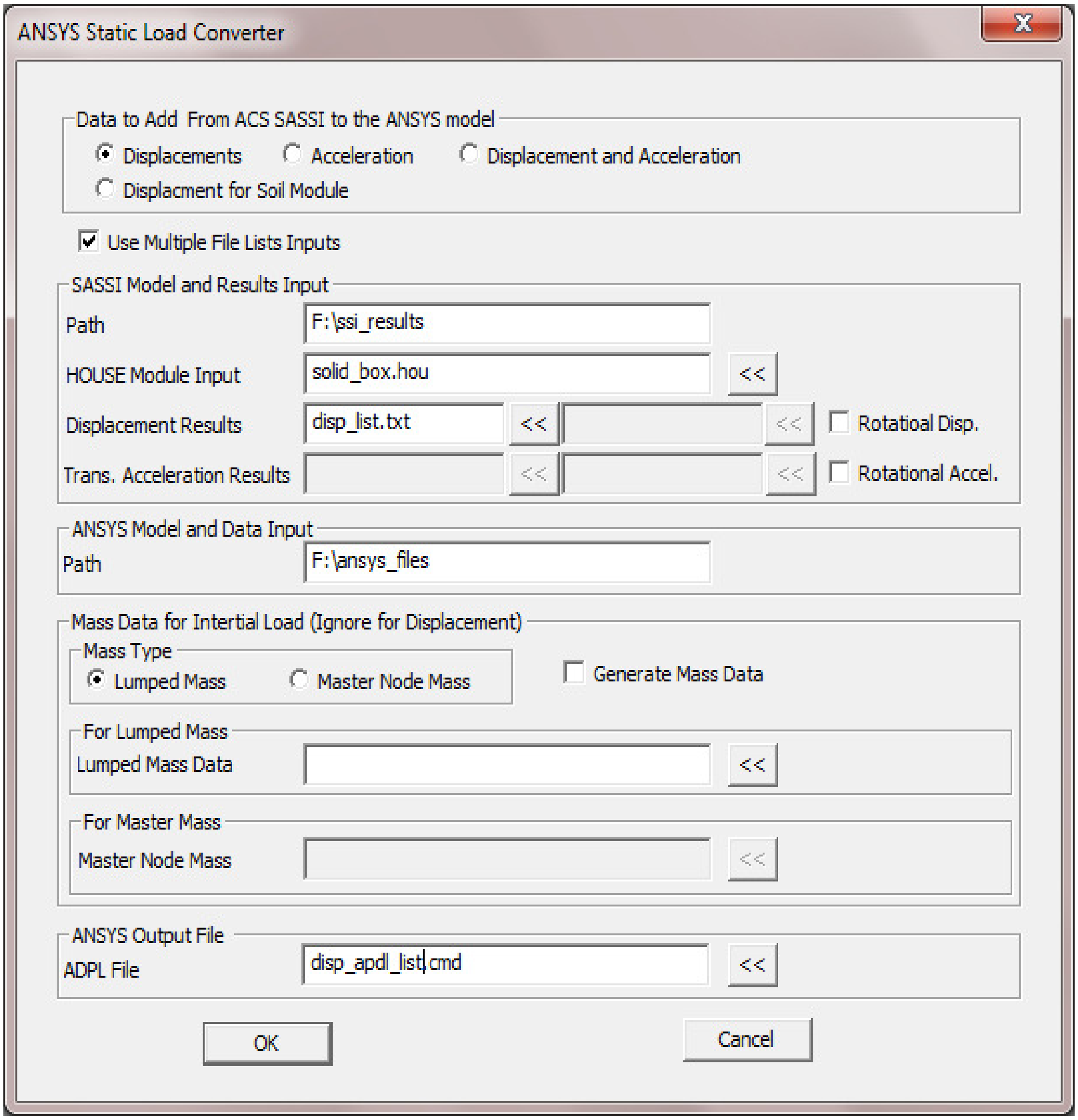

Figure 8: ANSYS Load Generator "Displacements" Option with Multiple Load Step Files

Generation of Nodal Mass Data for Computing Equivalent-Static Seismic Forces for "Acceleration" and "Displacement and Acceleration" Options

If the user wishes to apply the equivalent-static seismic forces on the structure, the user must generate the nodal mass data first. In order to generate the nodal mass data for inertial forces, the user should follow these steps shown below:

1. Check the "*Generate Mass Data*" box in the "*Mass Data for Inertia Load*" section. After checking that box, the program enters the nodal mass data generation mode regardless of what function the user chooses in the selection box. With this box checked, no seismic load

data is produced. Only the ANSYS APDL command file for generating nodal mass data will be generated. This file will be used later to generate mass data in given file for calculating seismic force loads on structure.

2. Select the appropriate nodal mass data type radio button, based on your plan and prepared work. There are two types of mass data for equivalent-static seismic forces. The *"Lumped Mass*" option will calculate the nodal mass data from the Coarse ANSYS model with "LUMPM, 1" setting. The "*Master Node Mass*" option will calculate the nodal mass data from the Detailed ANSYS model at the master nodes. The lumped mass option is selected by default. The user interface is shown in Figures 9 and 10 for "*Lumped Mass*" and "*Master Node Mass*", respectively.

3. If the user selects "*Lumped Mass*", then go to Step 4; If "*Master Node Mass*" is selected then, the go to Step 7.

4. Input the following parameters as shown in Figure 9:

   a. Enter the path of the ANSYS working folder in the group box marked "*ANSYS Model and Data Input"*, which would be "F:\ANSYS_Files" in our demonstration;

   b. Enter the file name "lumped_mass.dat" that will contain the lumped mass data in the input box marked "Lumped Mass Data". This file will be generated in Step 6 a by running the input APDL file.

   c. Enter the file name "get_lumped_mass.cmd" file that will contain the APDL command that will be used to generate the lumped masses in the input box marked "Lumped Mass Data".

5. Click the "OK" button. By this action, an APDL command file with the file name specified in Step 4.c, "get_lumped_mass.cmd", will be generated in ANSYS working folder.

6. Run the APDL input file and batch commands:

   a. <u>Execute the "get_lumped_mass.cmd" APDL command file</u> after loading the *Coarse* ANSYS model in the ANSYS to generate the lumped mass data file. This lumped mass data file can now be used to generate seismic forces.

7. Input the required parameters as shown in Figure 10:

   a. Enter in the text box next to *"Path"* the path name of the ACS SASSI results folder that was prepared in Stage 1, "F:\SSI_Results";

   b. Enter the HOUSE module input file "solid_box.hou" in the text box next to *"HOUSE Module Input"*

c. Enter the file name that contains the master nodal mass data, “master_mass.dat”. This file will be created by ANSYS in Step 9, which will be used for calculating the seismic forces for the Detailed ANSYS model;

d. Enter the file name that contains the APDL commands to get the master node mass data, “get_master_mass.cmd”. This file will be used to generate the master nodal mass data in the input box marked with “APDL” File. This file will be used to generate master mass data in Step 9.

8. Click the "OK" button to generate the APDL command file with the file specified in step 7.d, “get_master_mass.cmd”.

9. Generate the reduced nodal mass data for the *Detailed* ANSYS model for the selected master nodes (that are automatically selected to be all the nodes used to define the *Coarse* ANSYS model).

   The user must execute the APDL command file, “get_master_mass.cmd”, after loading ANSYS refined model in the ANSYS. The master mass data file, “master_mass.dat”, will now be generated. This file will be used in the seismic force calculations in related sections.

As mentioned earlier, the nodal mass data calculation procedure requires the ANSYS model has no displacement constraints, and is able to be used for modal analysis with option "LUMPM,1". If the ANSYS model has any *released nodal degrees of freedom, ANSYS will not be able to perform a modal analysis.* Therefore, the user must remove the nodal degree of freedom releases from the ANSYS model. This is a limitation imposed by ANSYS for the nodal mass calculation. This restriction on the *node release* conditions only affects the mass data generation phase. This does not preclude the use of the ANSYS models with released degrees of freedom for performing the equivalent-static analysis. To generate the mass data based on the ANSYS model with released degrees of freedom, the user has to perform some extra steps during the nodal mass generation data phase as outlined below:

1. The user needs to make a copy of the ANSYS database (.db file) to use only for the mass generation phase.
2. Remove any displacement constraints, and any released degrees of freedom from the ANSYS model. *This will not affect the final results of the equivalent-static analysis since this model is only used for the nodal mass data generation.*
3. Follow the same steps as described on the previous page from Steps 1 through 9 using the ANSYS model with no released degrees of freedom.
4. After the nodal mass generation is complete, then the unmodified ANSYS model containing released degrees of freedom should be used to perform the equivalent-static analysis.

“Acceleration” Option

The “Acceleration” option is selected by clicking the radio button for “*Acceleration*” in the group input box of “*Data to Add from ACS SASSI to the ANSYS model*”. This option is used to generate the seismic load input file using the nodal acceleration data and nodal mass data. The user has two options to compute nodal mass data, the lumped mass and master node mass. The user has the option to generate multiple seismic load files for different selected critical time steps in a single LOADGEN run by checking “*Use Multiple File Lists Inputs*”. *Please note that to use this option, the nodal mass data must be generated first using ANSYS, as outlined later in this section***.**

The ANSYS load generator window for the lumped mass selection is shown in Figures 11 and 12 and for the reduced mass selection in Figures 13 and 14. The user should input all required parameters. After clicking the “OK” button, the seismic forces are saved in the APDL input file, for example *“acc_load_822.cmd”*, as shown in the bottom input box that is saved in the ANSYS work folder.

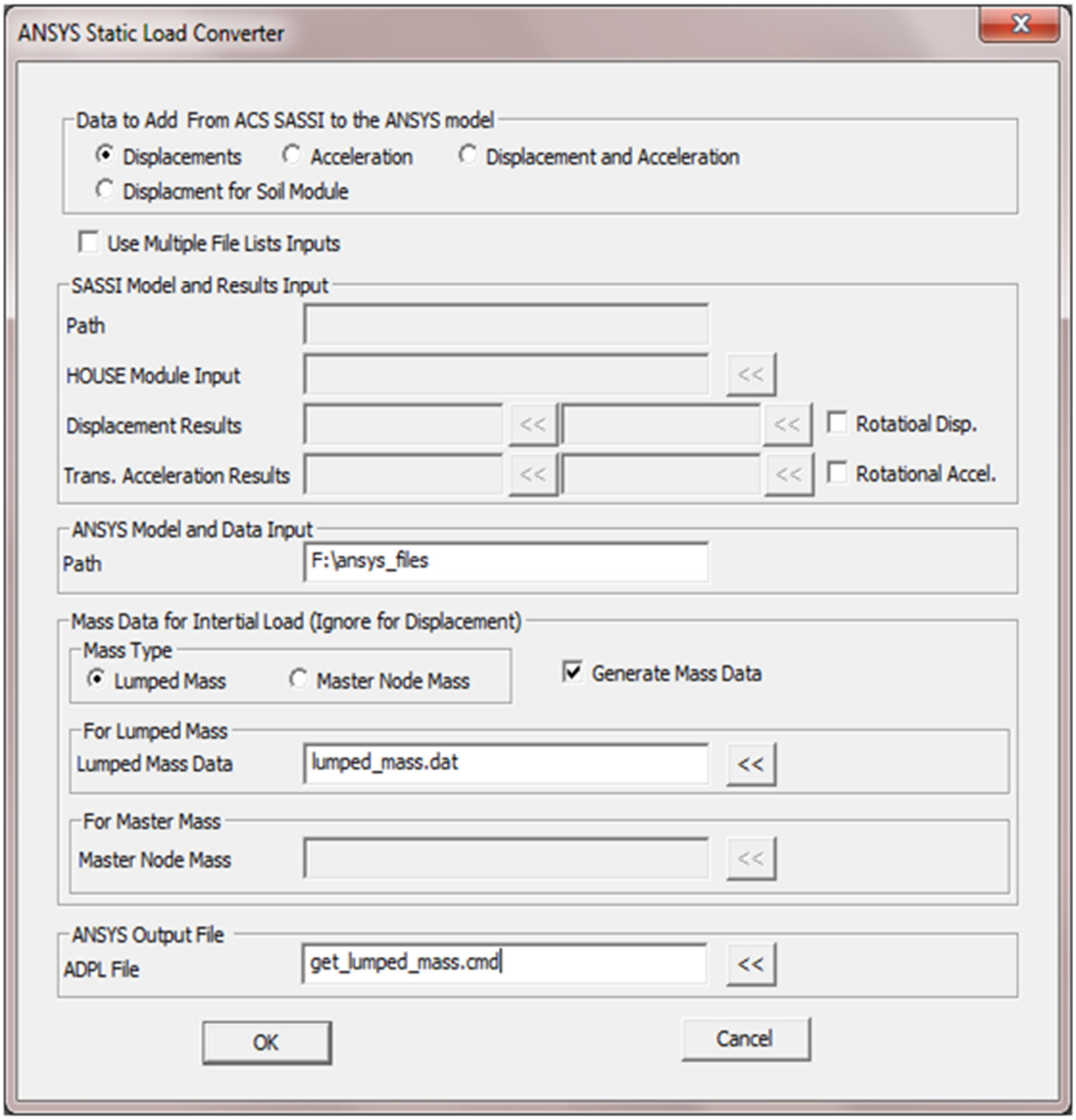

Figure 9: Generate Mass Data Using Lumped Mass Option

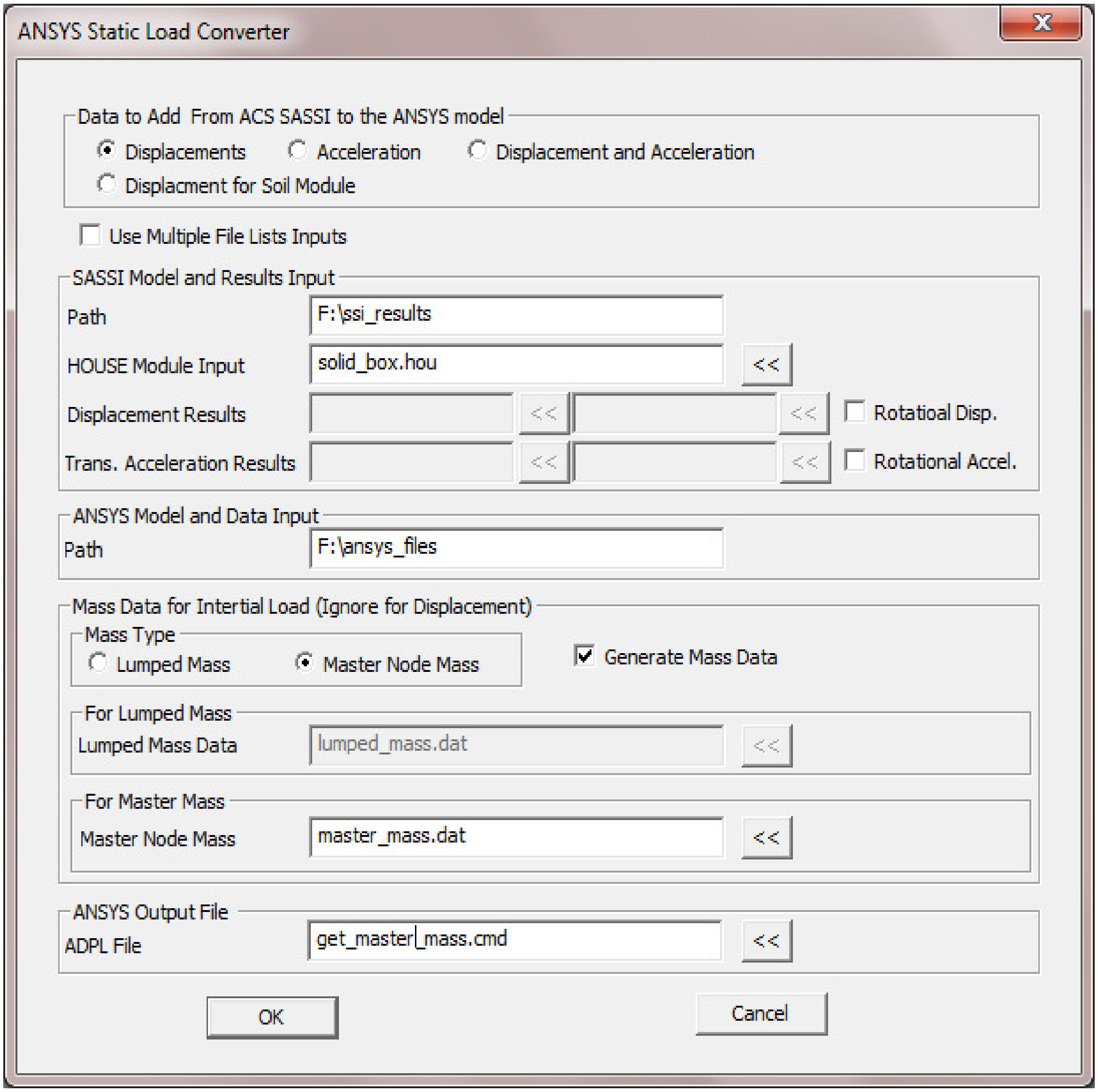

Figure 10: Generate Mass Data Using "Master Node Mass" Option

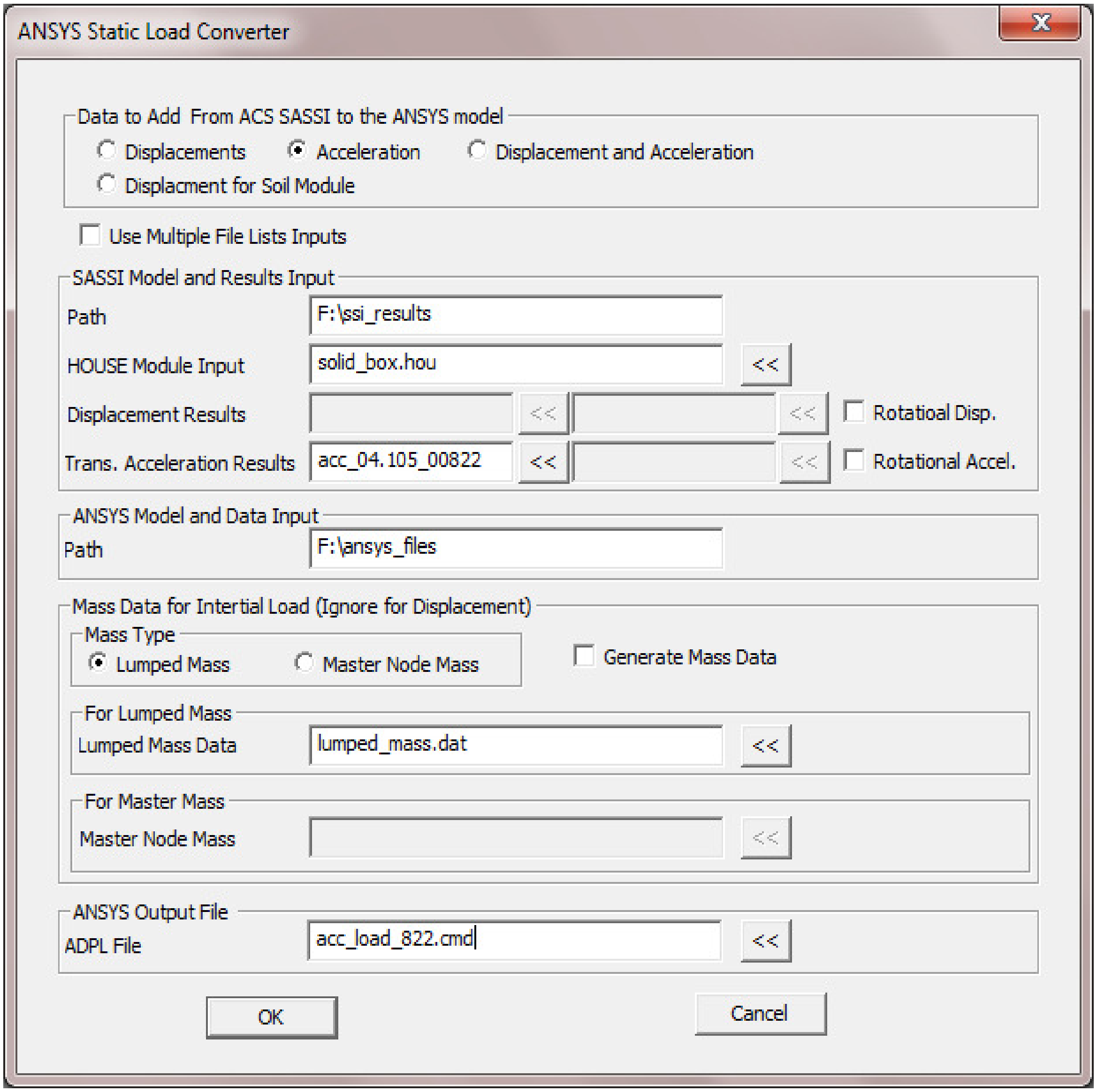

Figure 11: Acceleration Option with Lumped Mass for Single ANSYS Load file

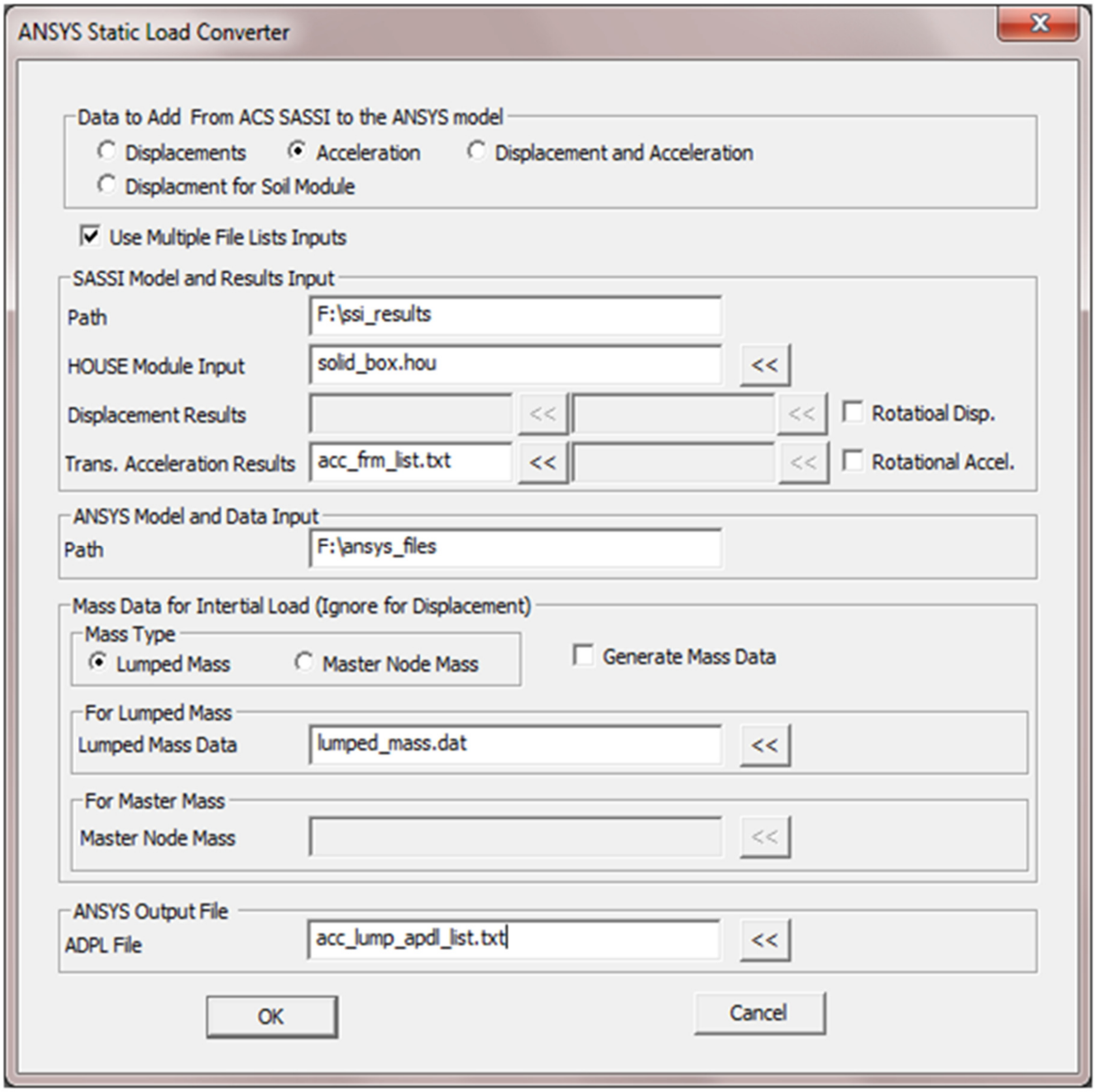

Figure 12: Acceleration Option with Lumped Mass for Multiple ANSYS Load Files

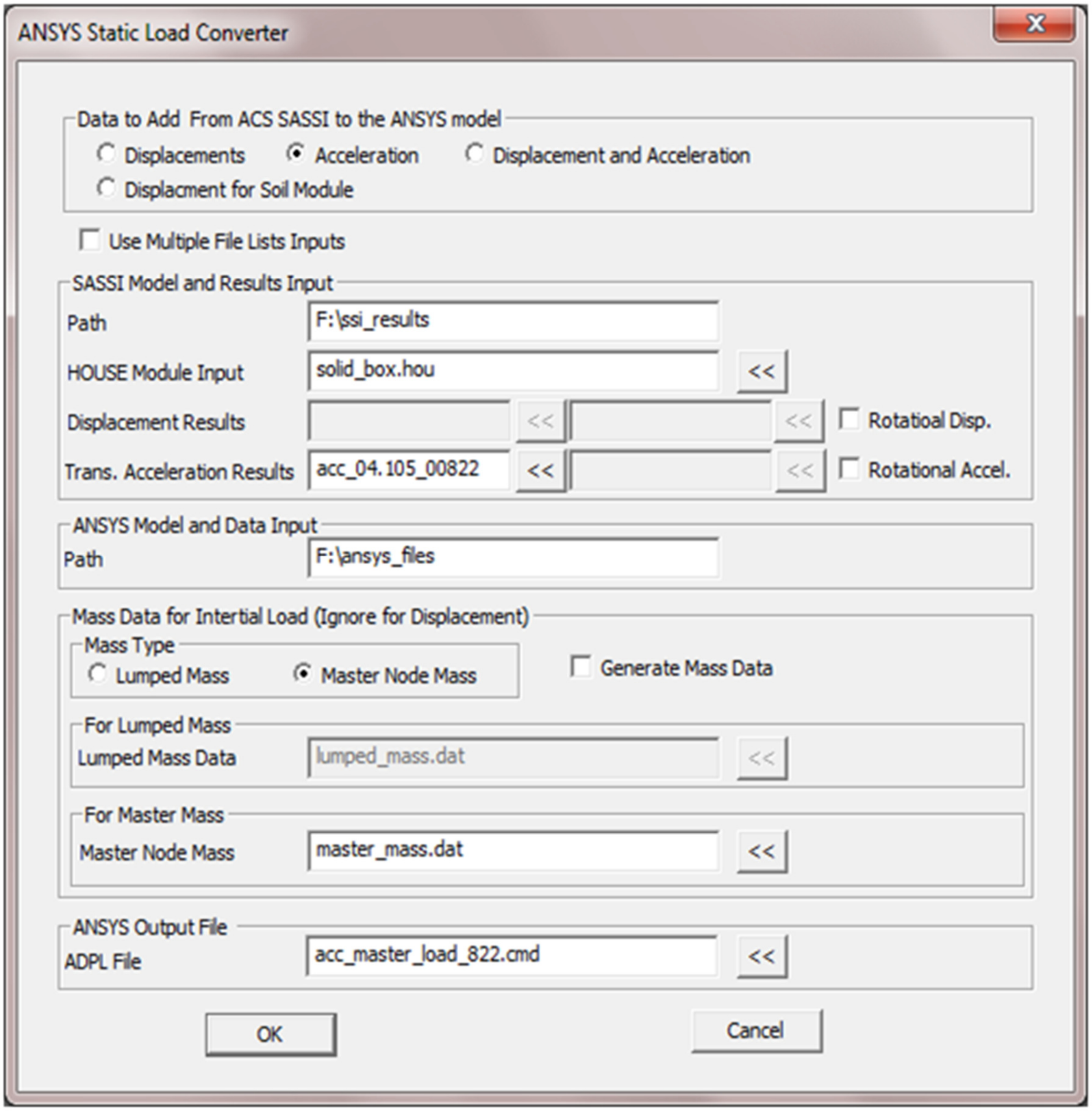

Figure 13: Acceleration Option with Master Node Mass for Single Load File

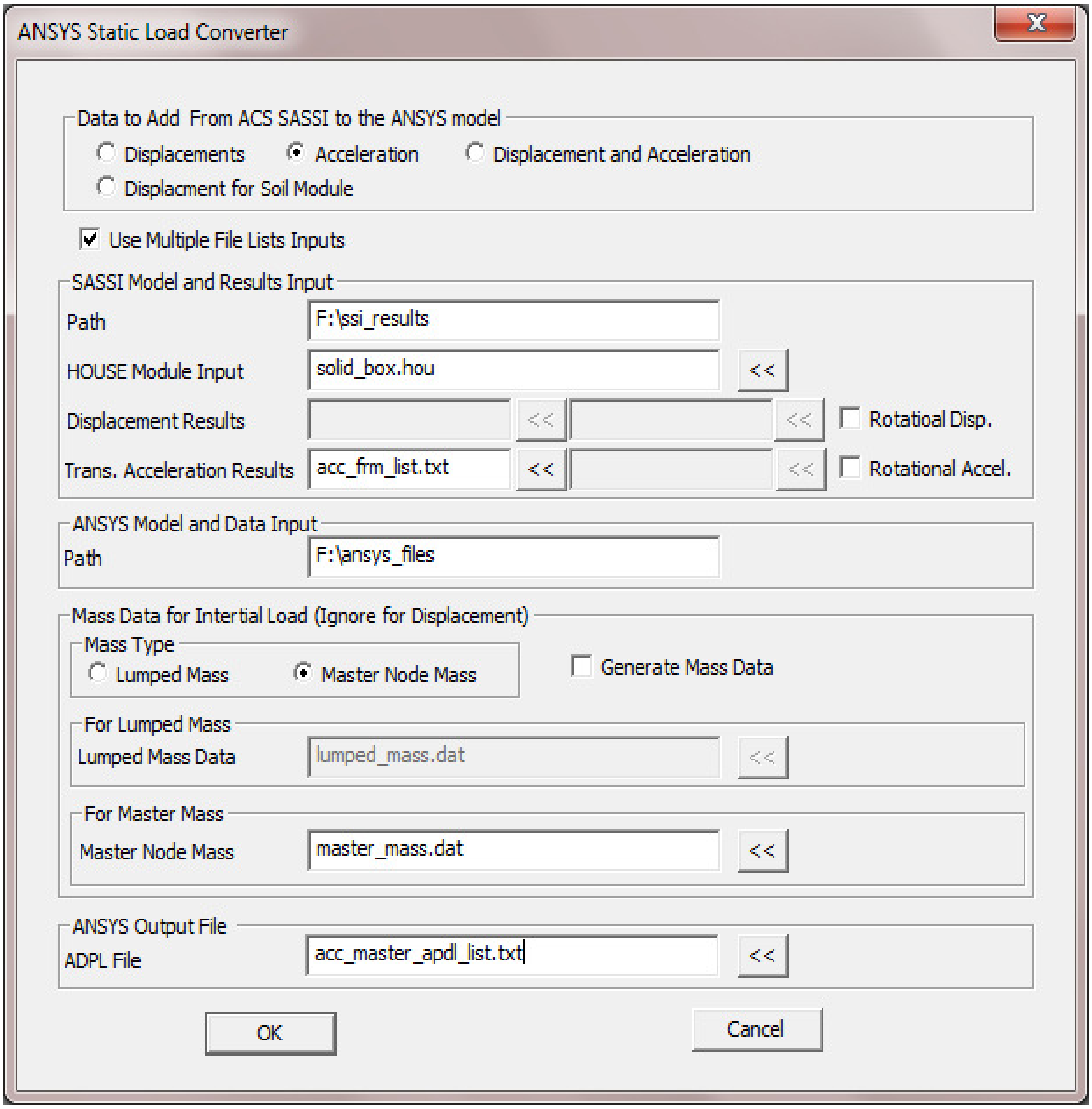

Figure 14: Acceleration Option with Master Node Mass and Multiple ANSYS Load files

It should be noted that for the *"Acceleration"* option, the user needs to apply the proper stiffness constraints at the foundation boundary nodes, usually by static soil springs placed at support nodes. However, the evaluation of these soil springs could be challenging, especially when the foundation is embedded, or with an arbitrary shape. A much more accurate solution for the foundation boundary conditions is to provide the relative displacements at support nodes with respect to free-field input motion at selected time steps.

"Displacement and Acceleration" Option

The "*Displacement and Acceleration*" option provides an appropriate set of boundary conditions for equivalent-static analyses. This option is designed to generate a mixed boundary condition

load file. This option will generate equivalent-static relative displacement boundary conditions at the support nodes (SSI interaction nodes of the ACS SASSI SSI model at the soil-foundation interface), and the equivalent-static seismic forces at all structural nodes.

The inputs depend on the selected nodal mass data option. The user can also select to generate multiple ANSYS load files by checking *"Use Multiple File Lists Inputs"* Figures 15 and 16 show the inputs using the "*Lumped Mass"* data option. Figures 17 and 18 show the inputs using the "*Master Node Mass*" data option. Once the seismic load file is generated, it can be applied to the *Detailed* ANSYS model as described for the other two loading options.

"Displacements for Soil Model" Option

This option is used to convert the ACS SASSI displacement results at the interaction nodes into the displacement BCs for the ANSYS soil model used to compute the seismic soil pressures on foundation walls. The input is the same as for the "Displacement" option described in earlier in this section. Figures 19 and 20 show the user's inputs for this option.

WARNING: *ONLY the SSI interaction nodes at the foundation-soil interface may be included in the model used by the SUBMODELER module*

By default the SUBMODELER module assumes that the ACS SASSI SSI model uses the Flexible Interface (FI) method with SSI interaction nodes defined only at the foundation-soil interface. If the seismic SSI analysis was performed using the Flexible Volume (FV) method or Flexible Interface with additional nodes on the excavation volume surface, then the HOUSE input file, must be modified to include only the SSI interaction nodes that are at the foundation-soil interface.

It should be noted that the SSI analysis can be done with either FI or FV SSI substructuring approaches, but for the seismic soil pressure computation using ANSYS, only the relative displacements with respect to free-field motion computed at the foundation-soil interface should be used. For this reason the .hou file copied in the *"SASSI Model and Results Input"* folder "F:\SSI_Results" should be modified to include interaction nodes only at the foundation-soil interface. This can be done by modifying the .pre file to define interaction nodes only at the foundation-soil interface. The modified .hou file that is produced by the AFWRITE command is then to be copied in the SSI results folder before the "Displacement for Soil Model" option is used.

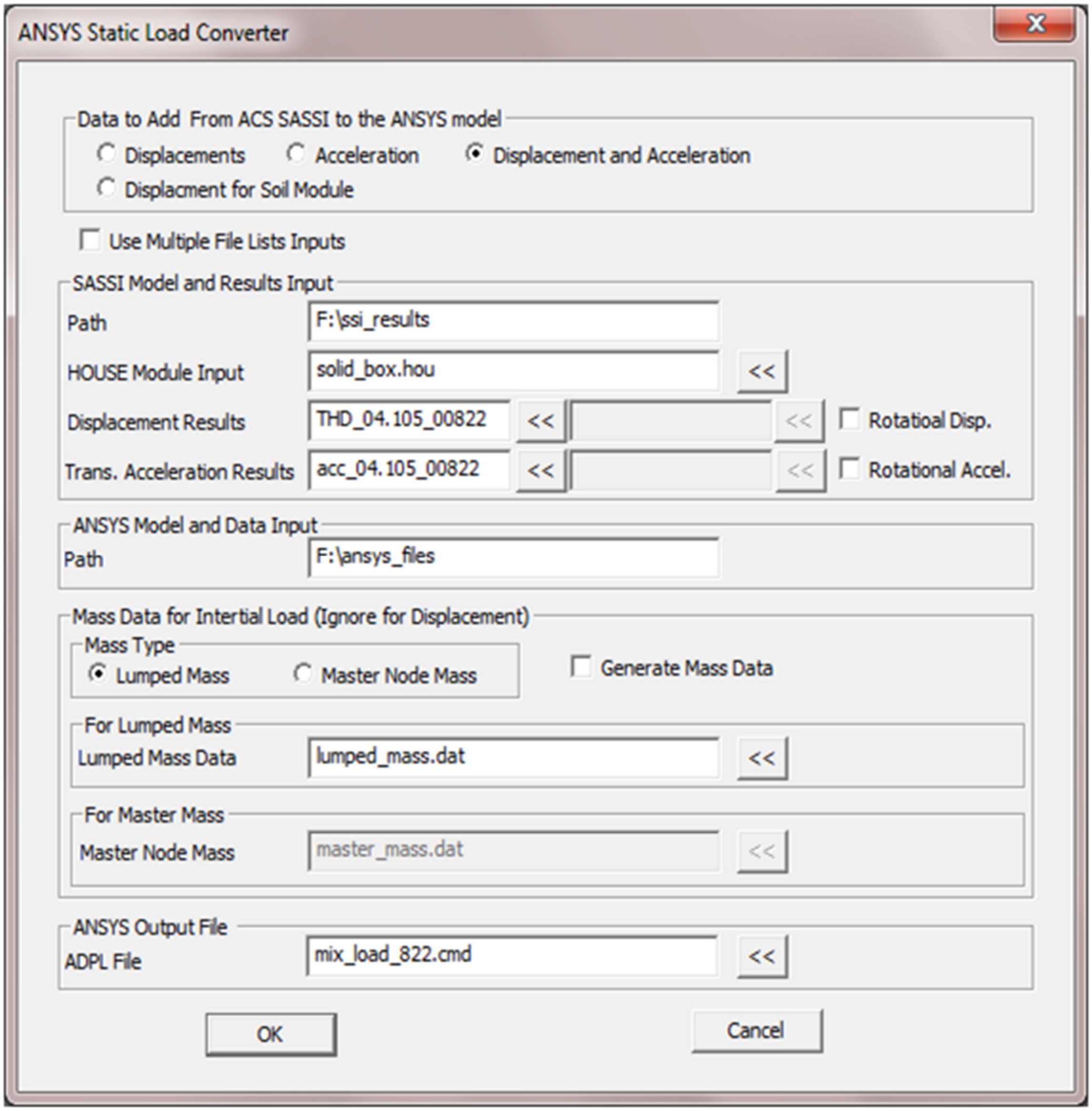

Figure 15: Displacement and Acceleration Option with Lumped Mass and Single ANSYS Load File

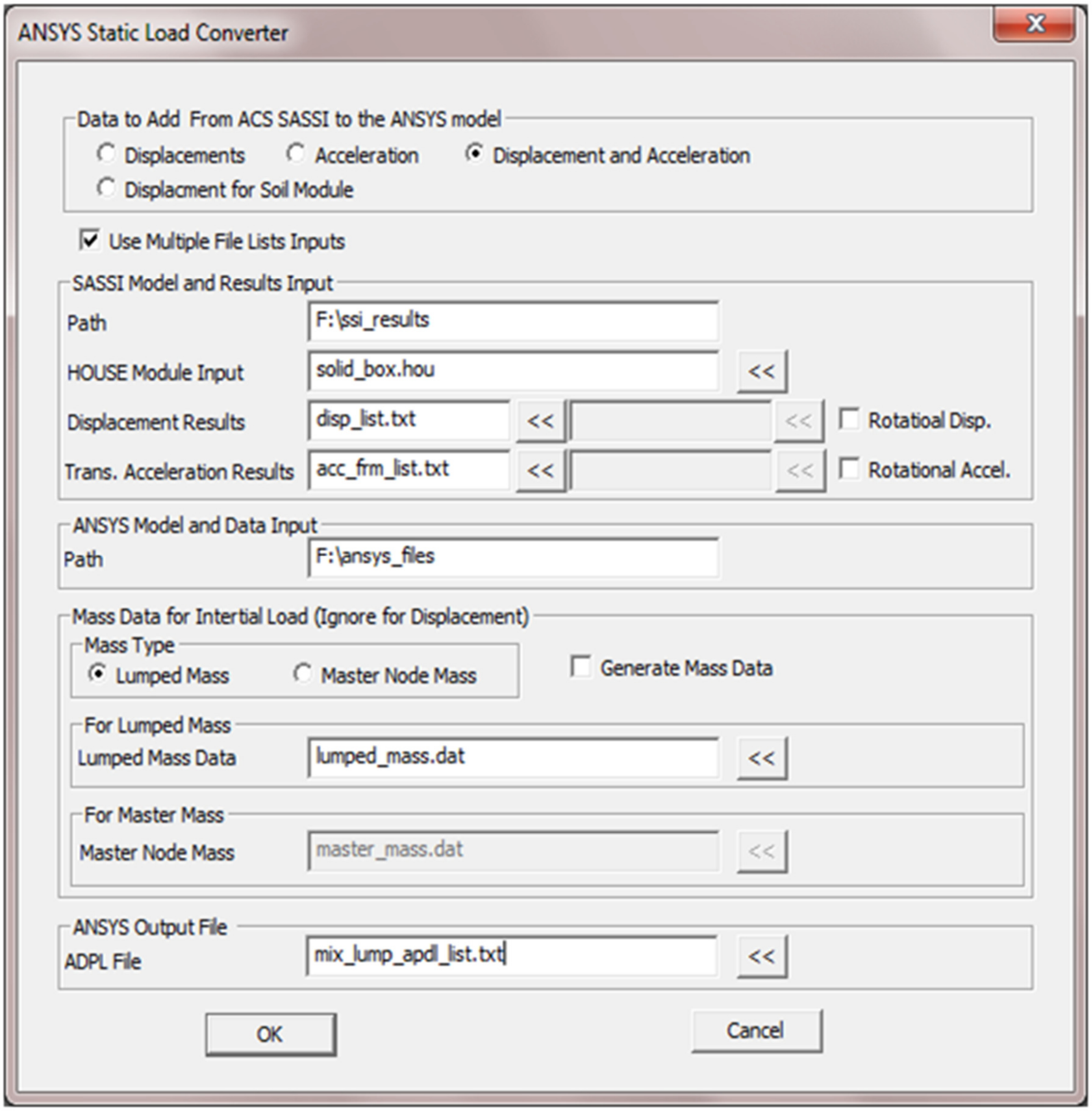

Figure 16: Displacement and Acceleration Option with Lumped Mass and Multiple ANSYS Load File

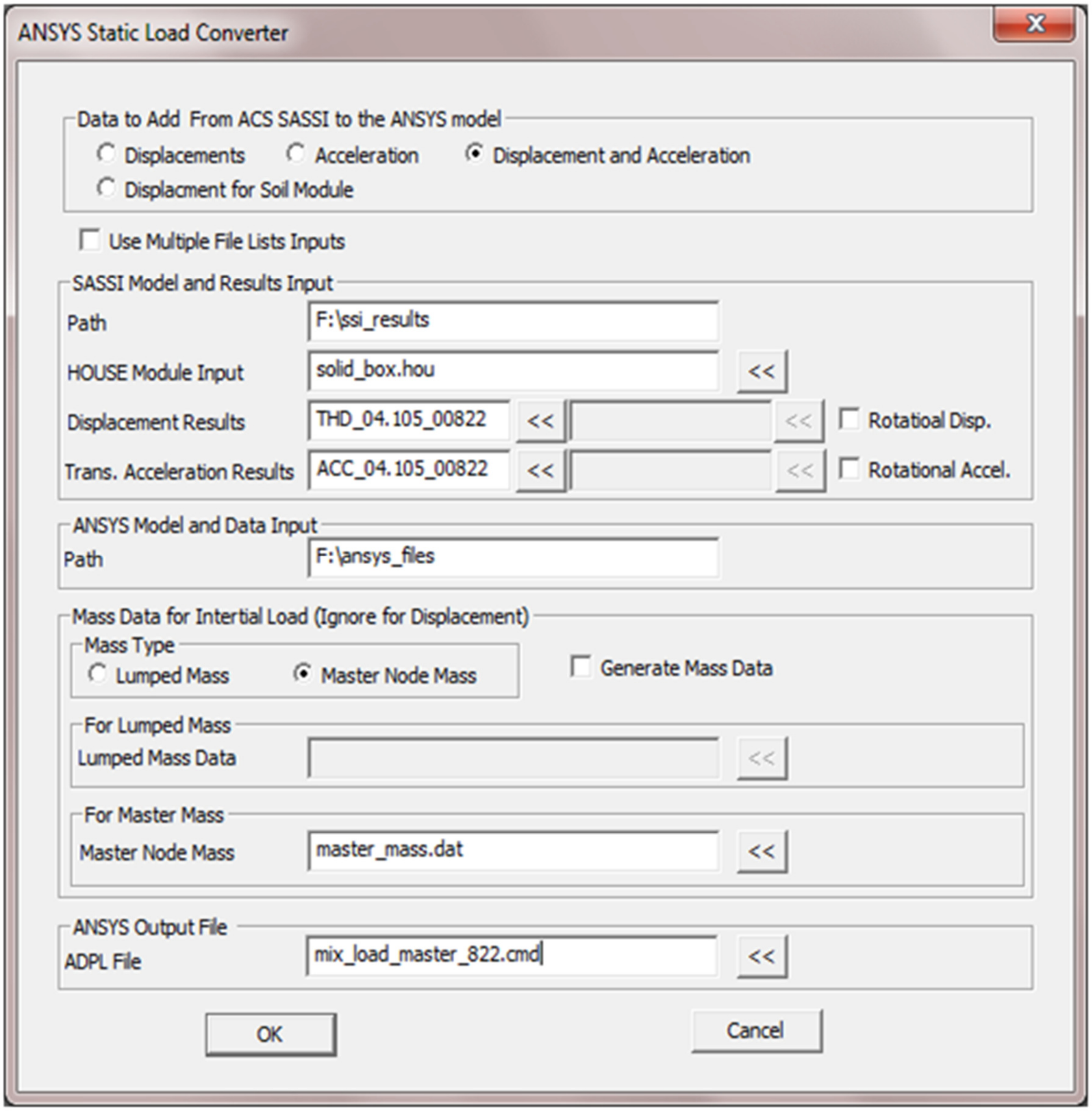

Figure 17: Displacement and Acceleration with Master Node Mass
and Single ANSYS Load File

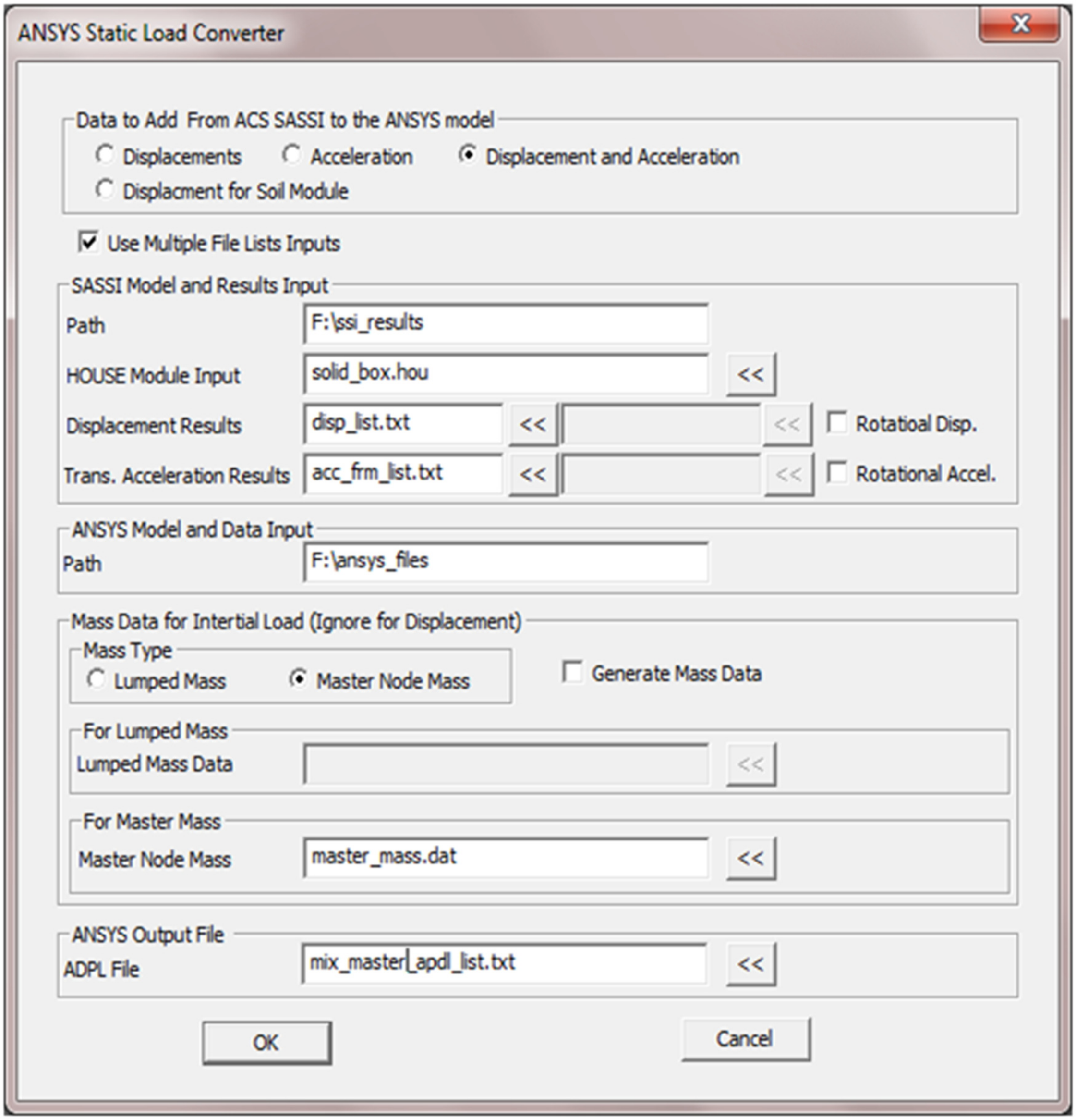

Figure 18: Displacement and Acceleration with Master Node Mass
and Multiple ANSYS Load Files

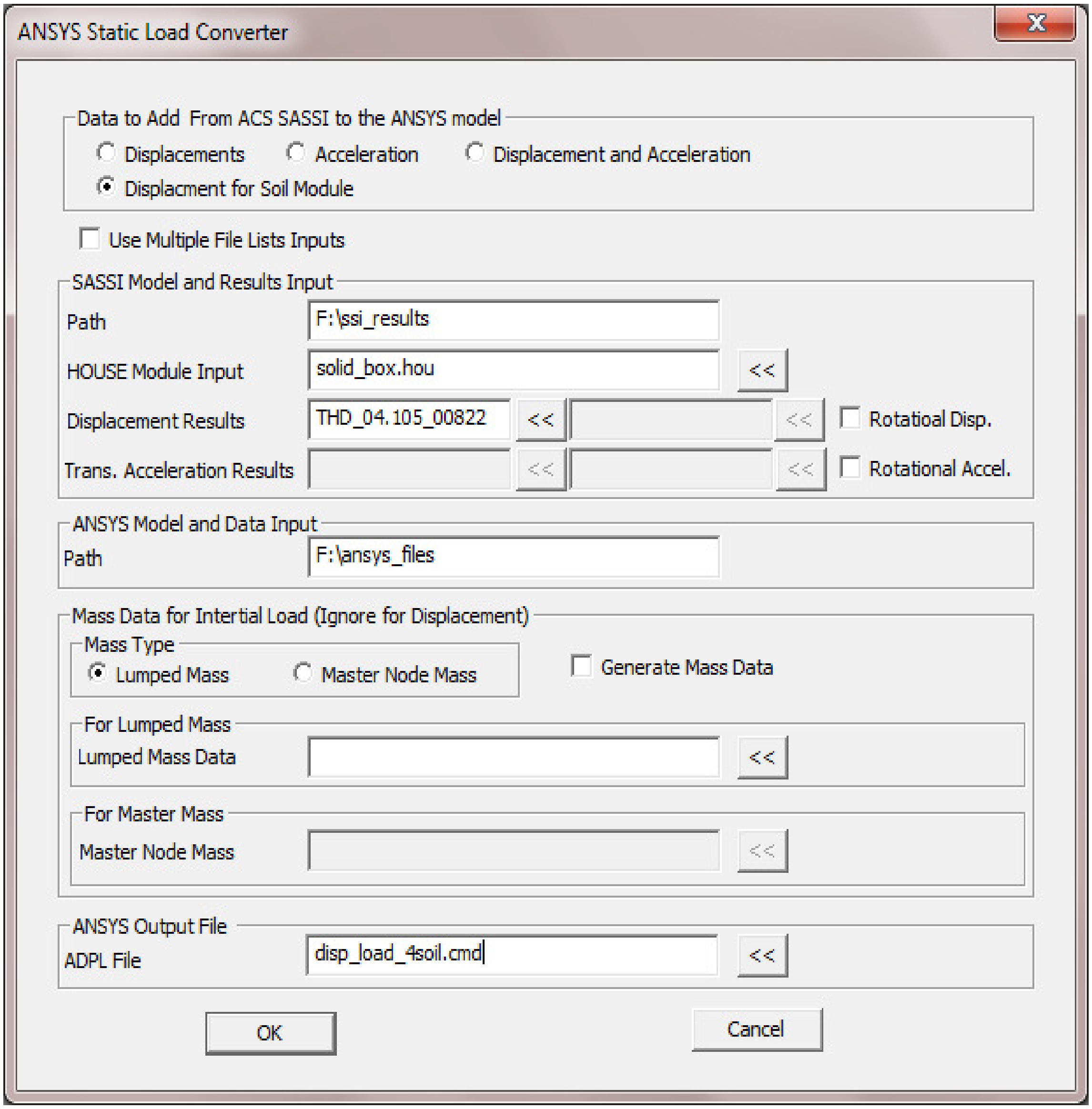

Figure 19: Displacement for Soil Model Option for Single ANSYS Load File

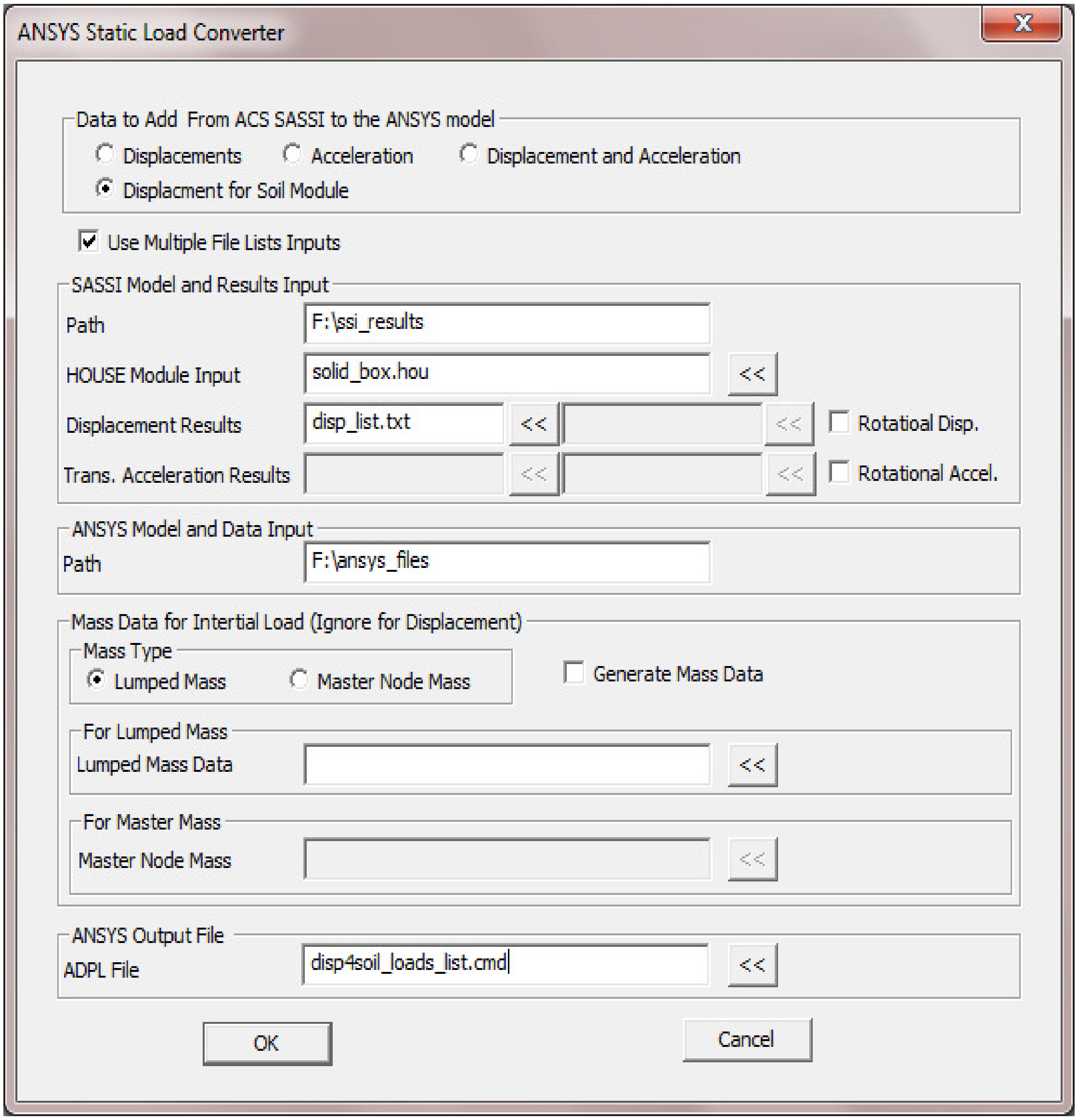

Figure 20: Displacement for Soil Model Option with Multiple ANSYS Load Files

Stage 2: Using LOADGEN for ANSYS Dynamic Analysis

To run this option, select *RUN→ "ANSYS Dynamic Load"* from the ACS SASSI MAIN menu. This will open the *"ANSYS Dynamic Load Converter"* window for generating the seismic loading and support boundary conditions for the ANSYS dynamic analysis load as shown in Figure 21.

Since the user has prepared all the necessary data in stage 1, the user just needs to fill the input boxes in *"ANSYS Dynamic Load Converter*" window with relevant items.

1. In the "Path" box in the *"SASSI Model and Results Input"* section, enter the path of the folder that contains the displacement frames, "F:\ssi_results"

2. In the *"HOUSE Module Input"* box, enter the file name of the .hou file, or browse to it by clicking on the arrow next to the box, "solid_box.hou"

3. In the "*Ground Acceleration File"* box, input the name of the acceleration file "ground_acce.txt". This file will be prepared by the user for the ANSYS dynamic analysis. The file format of the "ground_acce.txt" file is described in the next section.

4. In the "*Path*" box in the "ANSYS Model and Data Input" section, enter the path of the ANSYS file folder, "F:\ANSYS_Files"

5. In the *"Rayleigh Damping Coeff."* section, enter the Raleigh damping coefficients, alpha, beta, for the ANSYS dynamic analysis. The input data shown in Figure 21 corresponds roughly to the 5% viscous damping ratio in the low frequency range.

6. In the *"ANSYS Output File"* section, enter the name of the file that will contain the ANSYS APDL input commands for the dynamic analysis using the time-domain direct integration method. This is the file will be loaded in ANSYS.

7. Click "OK" button to generate all the dynamic step load files and file defined in step 6.

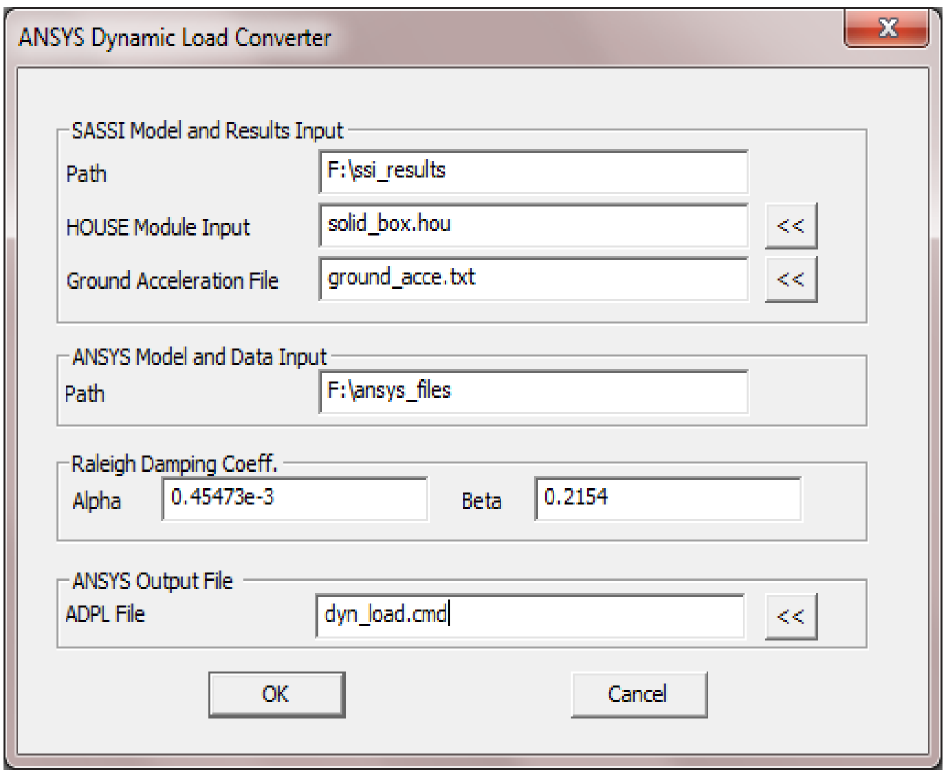

Figure 21: ANSYS Dynamic Load Generator Window

Note: During the generation of the dynamic step load files, the ANSYS dynamic load generator uses all the relative displacement data files with respect to the free-field motion from the SSI analysis results. The user should make sure these files have been copied to the ANSYS folder defined in the *"Path"* input box of *"SASSI Model and Results Input"* section.

For embedded foundations these relative displacements must be computed at different depth levels with respect to the free-field motion at the same depth levels. This is required since the free-field motions at different depths are different than the control motion, which is defined at a single location. Only for surface foundations under vertically propagating waves should these relative displacements be computed only with respect to the control motion.

After all files were generated, the user can perform the ANSYS dynamic analysis using the "/INPUT" command.

The "Ground Acceleration File" defines the kinematic SSI response rigid body acceleration fields computed during the SSI analysis. The acceleration history data is identical with ground acceleration only for surface foundations under vertically propagating waves. The file format is shown in Figures 22 and 23. The first line is the control data, the other next lines starting from the second line to the end of the file, contain the acceleration time history data for all six degree of freedom in space. Acceleration data should be input as units of g. LOADGEN will use the acceleration due to gravity from house to convert the ground acceleration into consistent units for ANSYS.

Line #1: Δt, ***NComp***, ***Xcg, Ycg, Zcg***

Δt – the time step size of the acceleration time history

NComp – the number of acceleration components, NComp can be 3 or 6 only; if NComp=3 that means just 3 translation acceleration will be used in the ANSYS dynamic analysis; if NComp=6 that means the both translation acceleration and rotation acceleration data will be used in the ANSYS dynamic analysis

Xcg, Ycg, Zcg – the rotation center of rotation acceleration in the global coordinate system.

Line #2: a_x,a_y,a_z if NComp=3 or
$a_x,a_y,a_z,a_x^r,a_y^r,a_z^r$ if NComp=6

where: a_x,a_y,a_z are the translation accelerations of three directions in global coordinate system,

a_x^r,a_y^r,a_z^r are the rotation accelerations around the three axes of the global coordinate system.

```
0.0050 3 30.0 40.0 30.0
 0.0000   0.0000   0.0000
 0.0000   0.0000   0.0000
 0.0000   0.0000   0.0000
 0.0000   0.0000   0.0000
 0.0000   0.0000   0.0000
 0.0001   0.0001   0.0001
   :        :        :
```

Figure 22: File Format of the Ground Acceleration File" with 3 DOF

```
0.0050  6  30.0  40.0   30.0
 0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
 0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
 0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
 0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
 0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
 0.0001    0.0001    0.0001    0.0001    0.0001    0.0001
 0.0001    0.0001    0.0001    0.0001    0.0001    0.0001
   :         :         :
   :         :         :
   :         :         :
-0.0081   -0.0081   -0.0081   -0.0081   -0.0081   -0.0081
-0.0091   -0.0091   -0.0091   -0.0091   -0.0091   -0.0091
-0.0104   -0.0104   -0.0104   -0.0104   -0.0104   -0.0104
-0.0121   -0.0121   -0.0121   -0.0121   -0.0121   -0.0121
-0.0141   -0.0141   -0.0141   -0.0141   -0.0141   -0.0141
```

Figure 23: File Format of the “Ground Acceleration File” with 6 DOF

3.5. ANSYS Equivalent-Static Analysis for Foundation Soil Pressures

This section describes the procedure used to perform an ANSYS equivalent-static seismic soil pressure analysis.

3.5.1. Linear Analysis Steps

- Perform SSI analysis using ACS SASSI
- Create ANSYS model input file using ANSYS model converter and load it in ANSYS
- Using ANSYS Load Generator (LOADGEN) create ANSYS load input files in APDL format
- Using ANSYS Soil Mesh Generator (SOILMESH) create soil model
- Convert soil model to ANSYS using SOILMESH ANSYS command and load the APDL soil model in ANSYS
- Load the APDL input with generated node displacements in ANSYS
- Solve using ANSYS

3.5.2. Nonlinear Analysis Steps

- Perform SSI analysis using ACS SASSI
- Create ANSYS structural model input file using the ANSYS model converter
- Load the ANSYS structure model
- Using ANSYS Load Generator (LOADGEN) create ANSYS load input files in APDL format
- Using ANSYS Soil Mesh Generator (SUBMODELER) create soil model grid, eventually including contact surfaces
- Create the ANSYS soil model using SUBMODELER *"ANSYS"* command
- Load the APDL soil model in ANSYS
- Load the APDL input file for seismic loads on structure in ANSYS
- Load the APDL soil model
- Run ANSYS for the integrated structure-surrounding soil model with or without contact surfaces at foundation-soil interface

A complete example for using Option A for computing seismic soil pressures on an embedded foundation including soil separation effects is shown in detail in the Demo 6 problem.

IMPORTANT NOTES:

Linear Analysis:

1) Only the soil model mesh is needed for performing linear soil pressure analysis. Once the soil mesh is generated by the SUBMODELER module, it can be loaded into ANSYS without the need to include the structural model. The *"Displacements for Soil Module"*

option creates a set of support node relative displacement inputs that can be applied directly to the soil model at foundation-soil interface nodes.

WARNING: *This option assumes by default the use of FI method with the interaction nodes distributed at the foundation-soil interface. If FV method is applied, or if FI method is used with additional interaction nodes, then, the .hou file has to be modified as described in Section 3.4.3.*

2) The user should apply appropriate boundary conditions on the soil model boundaries of the soil mesh. SUBMODELER automatically fixes all degrees of freedom at the bottom of the soil deposit. No boundary conditions are imposed on the lateral surface of the soil model.

Nonlinear Analysis:

3) For this case, both the structural and soil models are needed to perform the nonlinear seismic soil pressure analysis. These models can easily be merged in ANSYS simply by loading both .inp files for structure and soil deposit, one after the other. The created ANSYS .inp files using the ACS SASSI-ANSYS interface tools should be loaded in the following order:

    1. Input structural model
    2. Input load data
    3. Input soil model

   Loading the inputs in this order will ensure that none of the seismic loads are incorrectly placed on the soil elements (seismic loads are placed in the nearest neighbor nodes).

4) As with a linear soil pressure analysis, the user should apply appropriate boundary conditions on the soil model boundaries. SUBMODELER automatically fixes all degrees of freedom at the bottom of the soil deposit. No boundary conditions are imposed on lateral surface of the soil model. This avoids producing non-uniform displacement and stresses under gravity loads that are included in the nonlinear seismic soil pressure analysis.

5) If contact elements are generated, the user must input the number of the real constant to be used for the contact pair. This is the last argument in the SOILMESH command in SUBMODELER. The user should select a number that will not be used by another constant set for beams, shells, masses, springs, etc. Using a large number will usually prevent any overlap in real constant numbers.

6) The values for the real constants associated with the contact surface pair needs to be defined by the user. It is recommended that the user reads the ANSYS documentation for contact analysis prior to attempting a nonlinear soil pressure analysis due to the complexities of the analysis. Demo 6 includes recommended values for the real constant

parameters.

3.5.3. Automatic Generation of the Surrounding Soil Deposit

SUBMODELER is capable of automatically generating the FEA model of the surrounding soil. This surrounding soil model is then used for performing the ANSYS seismic soil pressure analysis, either linear or nonlinear.

In SUBMODELER, the user can control the soil model mesh refinement and extension in horizontal and vertical directions independently. This action will open the SUBMODELER window as shown in Figure 24. After the soil FEA model grid is generated using SOILMESH command, the ANSYS model is exported in the APDL input file format. Depending on the selected option, this ANSYS soil model can include contact surface elements for performing nonlinear soil pressure analysis to include foundation-soil separation effects and foundation sliding, if applicable.

The SUBMODELER module creates the surrounding soil model using SOLID elements as shown in Figure 2 in Section 3. The application of the SUBMODELER module to the ANSYS nonlinear seismic soil pressure analysis is described in detail in the demo problem 6.

To create the surrounding soil deposit model, the user must first load the ACS SASSI model input file (.pre file) that was used as input for ACS SASSI analysis into the SUBMODELER module. Then, from the SSI model excavated soil data, the SUBMODELER generates the new soil elements using SOILMESH command. Finally, using the "ANSYS" command in SUBMODELER, the new soil model is exported in the ANSYS APDL format. If the nonlinear analysis option is selected, then contact elements are included for modeling the foundation-soil contact interface. SUBMODELER generates contact pairs using the ANSYS TARGE170 and CONTA173 elements.

After loading the SSI model .pre file, the user should use the SOILMESH command in SUBMODELER to generate the soil deposit model. The initial SSI model (.pre file) is by default model number 0 (zero). The new generated soil deposit model will be the number selected in the first argument of the "SOILMESH" command. The user will need to activate the new model using the ACTM command before the ANSYS ADPL file can be exported using the SUBMODELER *"ANSYS"* command.

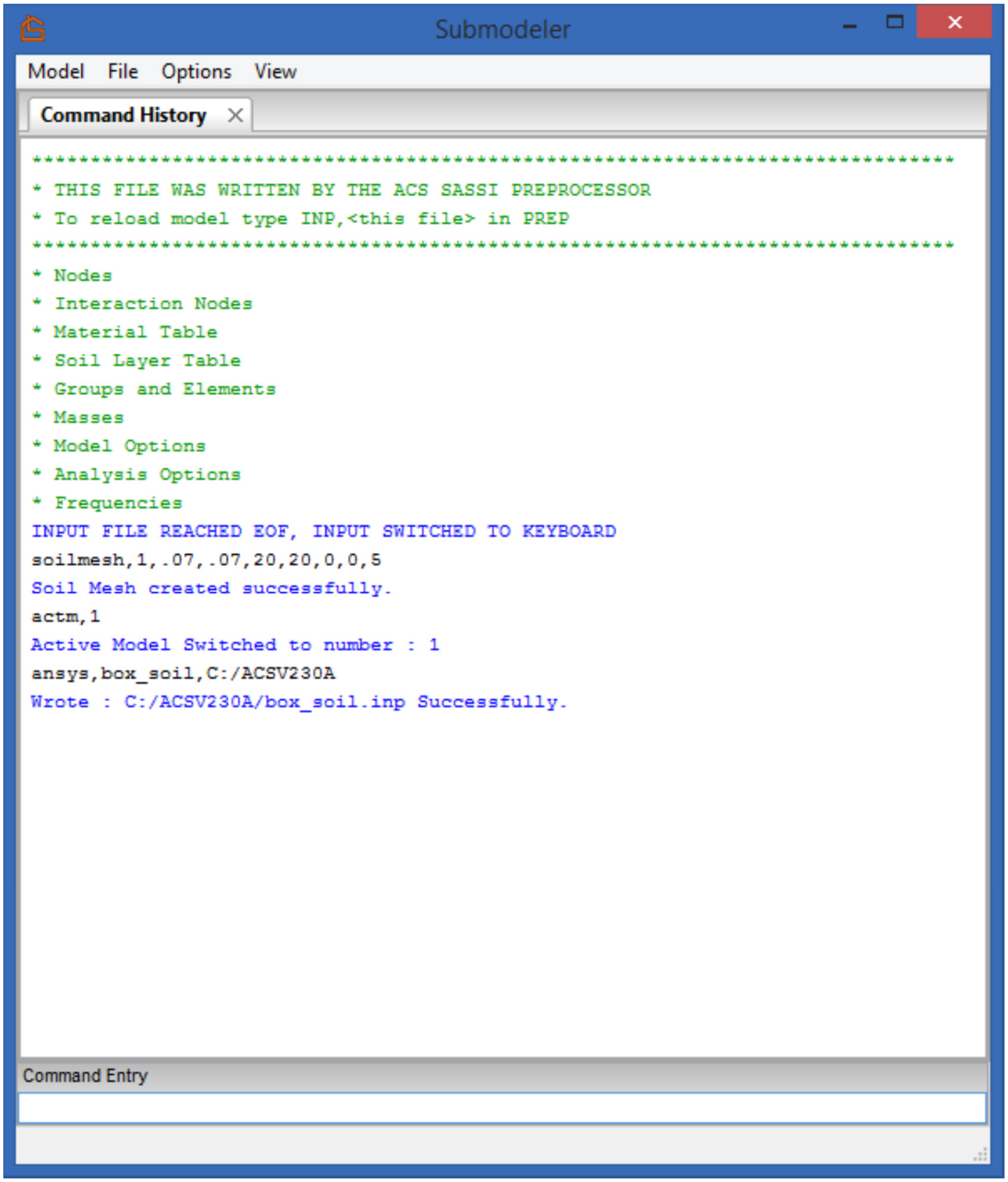

Figure 24 Application of the SUBMODELER Module SOILMESH, ACTM, and ANSYS Commands to Generate an ANSYS Surrounding Soil Model *

*Demo 6 illustrates in detail the use of the above commands for generating a soil deposit model.

4. OPTION "AA" OR "ADVANCED ANSYS"

The *Option AA* or Option Advanced ANSYS of the ACS SASSI-ANSYS integration capability enables the use of an ANSYS structural model for SSI analysis directly, without the need for converting the structural model to ACS SASSI. The ANSYS structural stiffness, mass and damping matrices are directly used by ACS SASSI for SSI analysis. The SSI relative displacements, absolute accelerations and response spectra for the ANSYS structural FEA model are fully computed within the ACS SASSI software. *Option A* should be used to transfer the SSI response motions at all time steps or selected critical steps as boundary conditions for the ANSYS superstructure model for computing structural stresses.

Option AA works with the fast-solver implementation only.

The Option AA was implemented by modifying the HOUSE fast-solver module and developing a new auxiliary program called SSI_ANSYS.exe. To make things simple, the new auxiliary program is wrapped inside an ANSYS APDL input file that will be run in ANSYS to produce the structural stiffness, mass and damping of the structure.

For embedded SSI models, both the structure FEA model and the excavated soil FEA model need to be developed in ANSYS. These ANSYS models are then loaded in the ACS SASSI SUBMODELER module and merged into a single SSI model.

For surface SSI models only the structure FEA model needs to be generated in ANSYS and transferred to ACS SASSI using SUBMODELER.

After the SSI model is created in SUBMODELER (including both the structure and excavated soil FEA models for embedded models, and only structure for surface models), it can be saved in the .pre format using the WRITE command, and in the .hou input format for the HOUSE module using the AFWRITE command. SUBMODELER can be also used to define the interaction nodes.

The modified HOUSE fast-solver module for Option AA is called HOUSEFSA. This modified HOUSE module is capable of reading the .hou input file created by SUBMODELER, even ANSYS element types that are incompatible with ACS SASSI element types are used.

WARNING: *It should be noted that the ACS SASSI SSI model saved in .pre or .hou formats it might include features from the ANSYS that are not compatible with the ACS SASSI finite element types. For example, if the PIPE or LINK elements are used in the ANSYS model, they will be automatically transformed into "dummy" BEAM elements by the SUBMODELER converter. Any .hou input file generated from the AFWRITE command that contains these "dummy" elements will also contain a flag to indicate that the input file should not be used directly in any SSI analysis, as it ONLY contains information on the model geometry. This will have no impact on the SSI analysis using Option AA, since the HOUSE model will use the ANSYS matrices directly without reading any element material or section information from the .hou file. While none of the matrices*

are formed from the .hou file that contains the "dummy" elements, this .hou file is still needed for the FE model configuration information including node coordinates, element groups, element connections which are needed for the ACS SASSI post-processing of the SSI results including structural animations.

4.1. **ANSYS APDL Procedure to Generate Structural Matrices**

Before the modified HOUSE fast-solver module for Option AA can be used, the ANSYS model matrices need to be extracted. The modified HOUSE module can then be used to read these matrices and convert them in the appropriate format for the ACS SASSI SSI analysis.

It should be noted that in Option AA, ANSYS models can include more sophisticated element types than the basic element types included in ACS SASSI. ANSYS Versions 14 or later are the only versions compatible with Option AA. The ANSYS model has to satisfy specific requirements as described below.

WARNING: *The ANSYS models to be used for the ACS SASSI Option AA analysis shall not include free nodes that are not connected to any elements. If any free nodes exist, user shall remove the free nodes and then, shall compress the node numbering.*

The ANSYS model shall not include any fixed nodal degrees of freedom with the "D" command.

The material damping ratio must to be defined by the BETD parameter.

The ANSYS model shall include only the following types of elements:

- SOLID element types: SOLID45 and SOLID185;
- SHELL element types: SHELL63 and SHELL181;
- BEAM element types: BEAM44 and BEAM188;
- PIPE element types: PIPE288;
- COMBIN element types: COMBIN14;
- Couple nodes (CP command) and Constraint equations (CE command)
- Multipoint constraint element types: MPC184 Rigid Link and/or Rigid Beam
- FLUID element types: FLUID80

To use the Option AA capabilities, the following steps must be performed prior to the SSI analysis run:

1) Generate the ANSYS model mass, stiffness and damping matrices using an APDL script that includes the execution of the SSI_ANSYS.exe program.

2) Create the HOUSEFSA module input file (.hou) using SUBMODELER

Steps 1 and 2 are described in Figure 25 and the text following below.

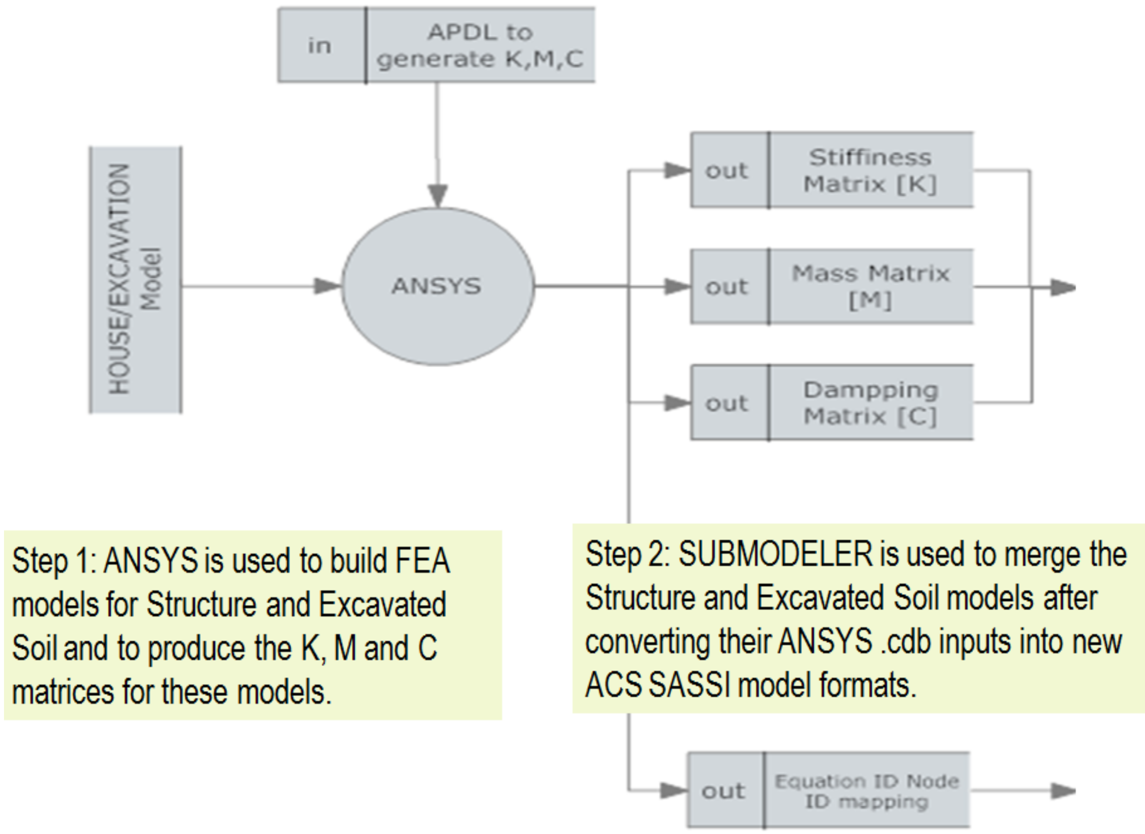

Figure 25 ANSYS Model Information Flow to ACS SASSI

To output the ANSYS model matrices, the user must run the ANSYS APDL macro called "gen_kmc.mac". This macro, as well as the SSI_ANSYS.exe program, is installed into the "ANSYS" subfolder in the installation directory. This APDL macro contains all the commands needed to generate the model mass, stiffness and damping matrices in the binary file format required by HOUSEFSA. The SSI_ANSYS.exe program is called as part of the APDL macro procedure. During the installation, the "ANSYS_MACROLIB" environmental variable is also set. This variable contains the path of the ANSYS subfolder in the installation directory.

If the SSI model is a surface model, then only the ANSYS structure model is needed. If the SSI model is an embedded model, then both the ANSYS structure and excavated soil models are needed.

There are a number of preliminary steps that are necessary to output the ANSYS model matrices to be used with the HOUSEFSA module in ACS SASSI. If the model is embedded, two matrix generation steps will be required; one for the structure, and one for the excavation volume. The steps are as follows:

1) Start ANSYS in the working directory with any job name

2) Load the structure model into ANSYS

3) Check if the ANSYS model meets the model requirements as described earlier (compatible element types, no D commands, damping defined with BETD)

4) Write the ANSYS model .cdb file using the ANSYS command "CDWrite,DB,*jobname*,cdb" for later use to create the input file for HOUSEFSA (.hou file)

5) Using the ANSYS APDL macro “gen_kmc.mac” generate the ANSYS model mass, stiffness and damping matrices in a binary file format that will be used as input files for the HOUSEFSA module. The following parameters are used for the structural model:

   - gen_kmc,'.',0,'.'

   After the macro has finished, the following files can be found in the ANSYS working directory.

   - coosk_r, cooski_r, coosm_r, coosmi_r, coosc_r, coosci_r, and Node2Equ_Stru.map

6) Save the database in ANSYS. If there is no excavation volume, skip to step 12. Steps 7 through 11 are only necessary for embedded models with excavation volumes.

7) Clear the database and start a new one. Set the job name to *jobname_excav*

8) Load the excavation model into ANSYS

9) Check if the ANSYS model meets the model requirements as described earlier (compatible element types, no D commands, damping defined with BETD)

10) Write the ANSYS model .cdb file using the ANSYS command "CDWrite,DB,*jobname*,cdb" for later use to create the input file for HOUSEFSA (.hou file)

11) Using the ANSYS APDL macro “gen_kmc.mac” generate the ANSYS model mass, stiffness and damping matrices in a binary file format that will be used as input files for the HOUSEFSA module. The following parameters are used for the structural model:

   - gen_kmc,'.',1,'.'

   After the macro has finished, the following files can be found in the ANSYS working directory.

   - cooek_r, cooeki_r, cooem_r, cooemi_r, cooec_r, coosei_r, and Node2Equ_Excv.map

12) Copy all the ANSYS model generated mass, stiffness and damping matrix files into the ACS SASSI working directory.

4.2. Using SUBMODELER to Generate the HOUSEFSA Input File

After the structure and the excavation ANSYS models are saved in the .cdb file format, they are loaded in the SUBMODELER module and, then converted into new ACS SASSI models. These new ACS SASSI models might include “dummy” parameters for the ANSYS elements that are not compatible, and therefore not convertible to the ACS SASSI element types. It should be noted that if the *Option AA* is selected, then the HOUSEFSA module will not use these “dummy” parameters and any of the converted elements to build the SSI model matrices, but will use the ANSYS model matrices directly, as described in the previous section. After the structure and excavation models are merged in SUBMODELER, the user can use the AFWRITE command for the HOUSE module to generate the .hou input file required by the HOUSEFSA module. The procedure of generating the .hou input file for the HOUSEFSA run is described in detail in the following sections.

4.2.1. **Surface SSI Models**

For surface SSI models only the ANSYS structure model .cdb file is needed. There are 12 steps to create the HOUSEFSA module input file (.hou).

1) Create a working directory in which the HOUSE input file (.hou) of the HOUSEFSA will be created using SUBMODELER
2) Copy the ANSYS .cdb file of the structure model into the new created SUBMODELER working directory
3) Start the SUBMODELER module
4) Convert the ANSYS structure model into the ACS SASSI format using SUBMODELER by selecting Model > Converters > ANSYS.cdb from the menu bar. A dialog box will open. In the dialog box input the following data
    a. In the "Input File Name" box input the ANSYS .cdb file name including path by typing or browsing ;
    b. In the "Output .pre File Name" box input the corresponding .pre file name including path (Optional step)
    c. In the "Save Converted Data to Model Number" box, enter the model ID number for the structure model. By default, this will save the converted model to model 0. The model ID number entered here should be an unused ID number in the current session. For this example, use 1 for the model ID
    d. In the "Enter Value for Gravity" box input the gravity acceleration; the gravity acceleration should be in the same units as the model either in ft/s2 or m/s2.
    e. Save the ANSYS model in ACS SASSI format by click the “Convert” button. The command history window shows the converting progress.
    f. Type “actm, *1*” to make the converted model the active model in SUBMODELER.
5) Define the ground elevation using SUBMODELER command "GroundElev" (for example “GroundElev, -10” would place the ground elevation at z = -10)

6) Generate the interaction nodes using SUBMODELER command "IntGen" (example," IntGen, 4" generates interaction nodes for surface models)

7) From the menu bar, select Options > Analysis. Select the HOUSE tab in the Analysis Options window. In this window, select the appropriate options for the SSI analysis, and any incoherency options that are to be used.

8) Check the box next to "ANSYS Model Input" and select "Surface"

9) In the Analysis Option window, select the AFWRITE tab. Check the box for HOUSE, so that the AFWRITE command will generate a new .hou input file

10) Assign a model name and full path to the model using the MDL command. (example "mdl,ModelName,J:\SSIproject\Model")

11) Write the .hou input file using the SUBMODELER command "AFWRITE" The HOUSEFSA input file will be generated in the file name assigned in step 10 with .hou as the extension

12) Copy the .hou input file to the ACS SASSI working directory that was created in the ANSYS matrix generation tutorial. This folder should also include the input files for the SITE, POINT, and ANALYS modules.

4.2.2. **Embedded SSI Models**

For embedded SSI models the .cdb files for both the ANSYS structure model and the ANSYS excavation model are needed. There are 19 steps to create the HOUSEFSA module input file (.hou).

1) Create a working directory in which the HOUSE input file (.hou) of the HOUSEFSA will be created using SUBMODELER

2) Copy the ANSYS .cdb files for the structure and excavation models into the SUBMODELER working directory;

3) Start SUBMODELER

4) Type "actm,1" to a switch to model ID number "1" which will be used for the structure model.

5) Convert the ANSYS structure model into the ACS SASSI format using SUBMODELER by selecting Model > Converters > ANSYS.cdb from the menu bar. A dialog box will open. In the dialog box input the following data

    a. In the "Input File Name" box input the ANSYS .cdb file name including path by typing or browsing ;

    b. In the "Output .pre File Name" box input the corresponding .pre file name including path (Optional step)

    c. In the "Save Converted Data to Model Number" box, enter the model ID number for the structure model. By default, this will save the converted model to model 0. The model ID number entered here should be an unused ID number in the current session. For this example, use 1 for the model ID

d. In the "Enter Value for Gravity" box input the gravity acceleration; the gravity acceleration should be in the same units as the model either in ft/s2 or m/s2.

e. Save the ANSYS model in ACS SASSI format by click the “Convert” button. The command history window shows the converting progress.

6) Define the ground elevation using SUBMODELER command "GroundElev" (for example “GroundElev, -10” would place the ground elevation at z = -10)

7) Type “actm, 2” to switch to model ID number “2” which will be used for the excavation model.

8) Convert the ANSYS excavation model into the ACS SASSI format using SUBMODELER by selecting Model > Converters > ANSYS.cdb from the menu bar. A dialog box will open. In the dialog box input the following data

   a. In the "Input File Name" box input the ANSYS .cdb file name including path by typing or browsing ;

   b. In the "Output .pre File Name" box input the corresponding .pre file name including path

   c. In the "Save Converted Data to Model Number" box, enter the model ID number for the structure model. By default, this will save the converted model to model 0. The model ID number entered here should be an unused ID number in the current session. For this example, use 1 for the model ID

   d. In the "Enter Value for Gravity" box input the gravity acceleration; the gravity acceleration should be in the same units as the model either in ft/s2 or m/s2.

   e. Save the ANSYS model in ACS SASSI format by click the “Convert” button. The command history window shows the converting progress.

9) Define the ground elevation using SUBMODELER command "GroundElev" (for example “GroundElev, -10” would place the ground elevation at z = -10)

10) Use the SUBMODELER command “EtypeGen” to define the solid elements in the excavation model as excavated soil elements by typing “EtypeGen, 2”

11) Use the SUBMODELER command “MergeSoil”, merge the structure model number 1 and the excavation model number 2 into a new combined model number 3

   a. Type “actm, 3” to switch to the model ID number “3” for the combined model.

   b. Use MergeSoil command to combine the structure and excavation models. The MergeSoil command also creates a mapping file required by the HOUSEFSA module. The mapping file provides the mapping between excavation model nodes and the corresponding combined model nodes. Note that this mapping file with extension .map should have a standard filename “modelname_excv.map”.

12) Generate the interaction nodes using SUBMODELER command "IntGen” (example, ”IntGen,1” generates interaction nodes for the Flexible Volume method)

13) From the menu bar, select Options > Analysis. Select the HOUSE tab in the Analysis Options window. In this window, select the appropriate options for the SSI analysis, and any incoherency options that are to be used.

14) Check the box next to "ANSYS Model Input" and select "Embedded"

15) In the Analysis Option window, select the AFWRITE tab. Check the box for HOUSE, so that the AFWRITE command will generate a new .hou input file

16) Assign a model name and full path to the model using the MDL command. (example "mdl,ModelName,J:\SSIproject\Model")

17) Write the .hou input file using the SUBMODELER command "AFWRITE" The HOUSEFSA input file will be generated in the file name assigned in step 16 with .hou as the extension

13) Copy the .hou input file to the ACS SASSI working directory that was created in the ANSYS matrix generation tutorial. This folder should also include the input files for the SITE, POINT, and ANALYS modules. The user must ensure that to the new .hou file (produced by AFRWRITE in Step 17) and the new map file (produced by MergeSoil in Step 11), as well as the coo… and node2equ… files generated with ANSYS as described in the previous section are in the ACS SASSI SSI analysis working directory.

***WARNING**: When the MERGESOIL command is used, the 1st model should be the structure model and the 2nd model should be the excavation volume. It is recommended to compress the node numbering for both ANSYS models if there are skipped nodes, so that the node numbering is compact for each model before the two models are merged in SUBMODELER.*

4.3. Running the HOUSEFSA Module

It should be noted that during the HOUSEFSA module execution, the extension name of input ".hou" will be changed to a new extension name ".hounew". This is due to the fact that during the HOUSEFSA module run, the model node numbering is automatically optimized to provide best numerical conditioning, storage use and computational speed. Thus, the new, final input filename for the HOUSEFSA module input is "modelname.hounew" filename not the initial "modelname.hou" filename generated using SUBMODELER. HOUSEFSA also produces a "modelname.map" file that provides the node mapping between the original SSI model created by SUBMODELER and the new optimized SSI model produced by HOUSEFSA. It should be noted that the FILE4 or "modelname.n4" file, and the COOSK and COOSM files produced by HOUSEFSA correspond to the optimized input model, not the original input model. Thus, the user should use the node mapping provided in this "modelname.map" file to extract the SSI analysis results at the correct locations when using the MOTION and RELDISP modules. The output node numbers will correspond to the node numbers of the optimized SSI model included in the "modelname.hounew" file.

The Demo 7 problem illustrates in detail two examples of using the ACS SASSI-ANSYS Interface in Option AA for a surface model and an embedded model.

